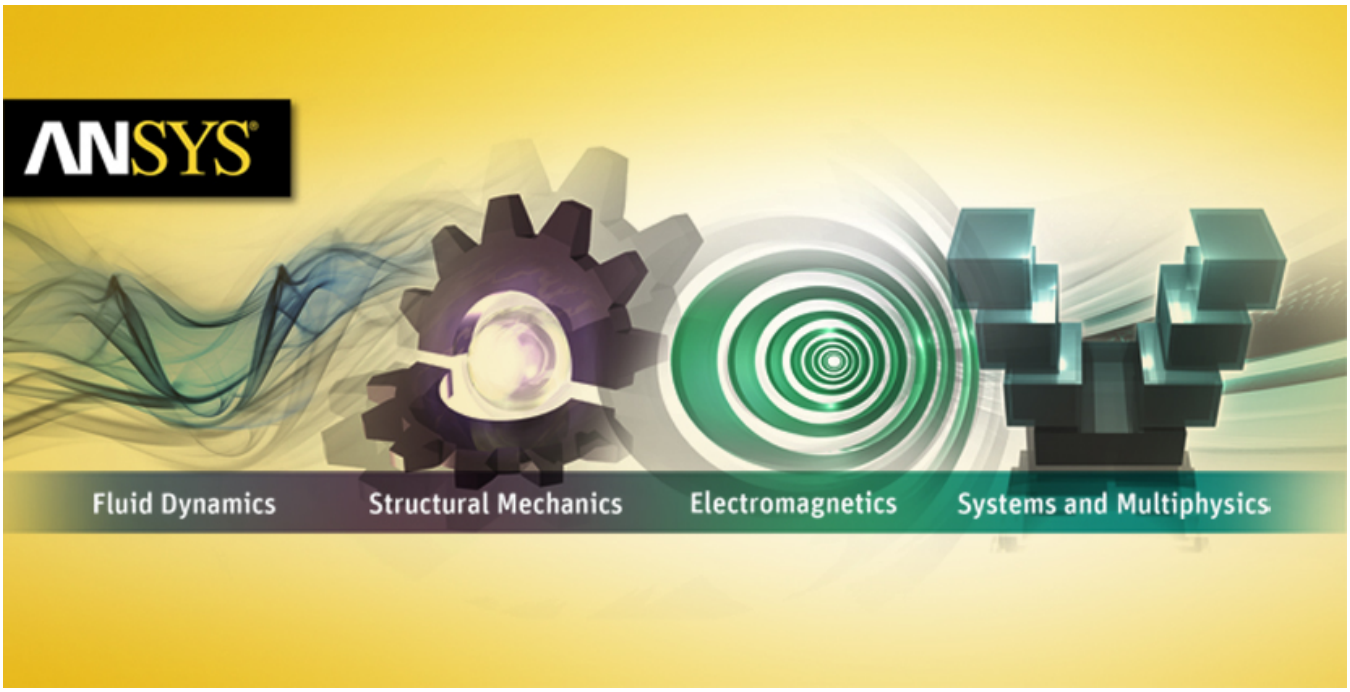


ANSYS ICEM CFD Help Manual

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

ANSYS ICEM CFD 15.0
November 2013

ANSYS, Inc. is certified to ISO 9001:2008.

Copyright and Trademark Information

© 2013 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Main Menu Area 1
File Menu 1
Project Management Options 2
Entity Management Options 4
Geometry 5
Mesh 7
Blocking 9
Attributes 11
Parameters 12
Cartesian 13
Import/Export Options 13
Import Model 14
Import Geometry 22
Faceted 23
ICEM CFD Mesh 23
Nastran 23
Patran 23
STL 24
VRML 24
Legacy 24
Acis 26
CATIA V4 27
DWG 29
GEMS 30
IDI 30
ParaSolid 31
Rhino 3DM 32
Plot3d 33
STEP / IGES 34
Formatted Point Data 35
Reference Geometry 36
Import Mesh 37
From ANSYS 37
From Abaqus 37
From CFX 37
From CGNS 39
From Fluent 39
From LS-DYNA 39
From Nastran 39
From Patran 40
From Plot3d 40
From Starcd 40
From STL 40
From TecPlot 41
From Vectis 41
Export Geometry 41
To IGES 42
To Parasolid 42
To Rhino 3DM 43
To STL 44

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export Mesh 44
To Abaqus 45
To ANSYS 45
To Autodyn 46
To Exodus 46
To Ideas 47
To LS-DYNA 47
To Nastran 47
To Patran 48
To Radioss 48
To FE Modeler 48
Write STL file 49
Spectral Elements 49
Replay Scripts 49
Exit 53
Edit Menu 53
Undo 53
Redo 54
Clear Undo 54
Shell 54
Facets > Mesh 54
Mesh > Facets 54
Struct Mesh > CAD Surfaces 54
Struct Mesh > Unstruct Mesh 54
Struct Mesh > Super Domain 54
Shrink Tetin File 55
View Menu 55
Fit 55
Box Zoom 56
Top 56
Bottom 56
Left 56
Right 56
Front 56
Back 56
Isometric 56
View Control 56
Save Picture 57
Mirror and Replicates 59
Annotation 60
Add in the current window 60
Modify by selecting 61
Pick and move 61
Pick and remove 61
Reset mouse 61
Add Marker 62
Clear Markers 62
Mesh Cut Plane 62
Info Menu 64
Geometry Info 65
Surface Area 65
Frontal Area 65

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curve Length 65
Curve Direction 65
Mesh Info 65
Mesh Area/Volume 66
Element Info 66
Node Info 67
Element Type/ Property Info 67
Toolbox 67
Project File 67
Domain File 67
Mesh Report 67
Settings Menu 69
General 70
Product 71
Display 73
Speed 78
Memory 78
Lighting 79
Background Style 80
Mouse Bindings/Spaceball 81
Selection 83
Remote 85
Model/Units 85
Geometry Options 88
Meshing 90
Hexa Meshing 90
Quality/Histogram Info 94
Edge Info 96
Solver 96
Reset 97
Help Menu 98
Help Topics 98
Tutorial Manual 98
User's Manual 98
Programmer's Guide 98
Output Interfaces 99
Installation & Licensing Guide 99
What's New 99
Show Customer Number 99
About ANSYS ICEM CFD 99
Graphical Main Menu, Utilities and Display Options 99
Main Menu Icons 100
Utilities Icons 102
Display Management Icons 104
Selecting Entities, Keyboard and Mouse Functions 107
Selection Options 107
Selection Mode Keymap 107
Location Selection Toolbar 111
Geometry Selection Toolbar 112
Mesh Selection Toolbar 114
Blocking Selection Toolbars 116
Density Selection Toolbar 119

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hotkeys 120
Geometry 121
Blocking 123
Edit Mesh 124
Spaceball and Mouse Binding 125
Display Tree 127
Model 128
Geometry 128
Geometry Subsets Options 129
Modify Geometry Subset Options 131
Geometry Points Options 133
Geometry Curves Options 134
Geometry Surfaces Options 136
Geometry Bodies Options 138
Geometry Densities Options 139
Mesh 140
Mesh Display Options 141
Mesh Subsets 145
Modify Mesh Subsets 148
Mesh Points 154
Mesh Lines 154
Mesh Shells 154
Mesh Volumes 158
Blocking 158
Blocking Options 159
Blocking Subset Options 160
Modify Subsets Options 161
Vertices 162
Edges 163
Faces 164
Blocks 164
Pre-Mesh 166
Topology 171
Local Coordinate Systems 171
Element Properties 172
Connectors 173
Constrained Nodes 173
Contacts 174
Displacements 175
Loads 176
Material Properties 177
Rigid Walls 177
Single Surface Contacts 178
Temperatures 178
Velocities 179
Parts 180
Parts Display Options 180
Create Part 180
Create Assembly 182
More Part Display Options 185
Parameters 190
Subcases 191

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry 193
Create Point 193
Screen Select 194
Explicit Coordinates 194
Base Point and Delta 195
Center of 3 Points/Arc 195
Based on 2 Locations 195
Curve Ends 196
Curve-Curve Intersection 196
Parameter along a Curve 197
Project Point to Curve 197
Project Point to Surface 197
Create/Modify Curve 197
From Points 198
Arc from 3 Points 198
Circle from Center and 2 Points 198
Surface Parameters 199
Surface-Surface Intersection 200
Project Curve on Surface 200
Segment Curve 201
Concatenate/Reapproximate Curves 201
Extract Curves from Surfaces 202
Modify Curves 202
Create Midline 206
Create Section Curves 206
Create/Modify Surface 207
From Curves 207
Curve Driven 208
Sweep Surface 208
Surface of Revolution 209
Loft Surface over Several Curves 209
Offset Surface 209
Midsurface 210
Segment/Trim Surface 213
Merge/Reapproximate Surfaces 213
Untrim Surface 214
Create Curtain Surface 214
Extend Surface 215
Geometry Simplification 217
Standard Shapes 221
Create Body 224
By Topology 225
By Material Point 226
Create/Modify Faceted 227
Create/Edit Faceted Curves 227
Convert from Bspline 228
Create Curve 228
Move Nodes 228
Merge Nodes 229
Create Segments 229
Delete Segments 229
Split Segment 229

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Restrict Segment 229
Move to New Curve 229
Move to Existing Curve 229
Surfaces 229
Convert from Bspline 230
Coarsen Surface 231
Create New Surface 231
Merge Edges 231
Split Edge 232
Swap Edge 232
Move Nodes 233
Merge Nodes 233
Create Triangles 234
Delete Triangles 234
Split Triangles 235
Delete Non-Selected Triangles 236
Move to New Surface 236
Move to Existing Surface 236
Merge Surfaces 236
Faceted Cleanup 236
Align Edge to Curve 237
Close Faceted Holes 237
Trim By Screen 238
Trim By Surface Selection 238
Repair Surface 238
Create Character Curve 239
Repair Geometry 241
Build Topology 242
Check Geometry 245
Close Holes 246
Remove Holes 246
Stitch/Match Edges 247
Using Stitch/Match Edges for Y-Junctions 249
Split Folded Surfaces 250
Adjust Varying Thickness 250
From Solid Method 250
Specify Corners Method 251
Find Surfaces Without Thickness Assigned 251
Make Normals Consistent 251
Feature Detect Bolt Holes 252
Feature Detect Buttons 254
Feature Detect Fillets 255
Transform Geometry 256
Translation 257
Rotation 257
Mirror Geometry 258
Scale Geometry 258
Translate and Rotate 258
Restore Dormant Entities 259
Delete Point 259
Delete Curve 260
Delete Surface 260

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete Body 260
Delete Any Entity 260
Mesh 261
Global Mesh Setup 261
Global Mesh Size 267
Shell Meshing Parameters 270
Autoblock Options 272
Patch Dependent Options 273
Patch Independent Options 280
Shrinkwrap Options 280
Volume Meshing Parameters 281
Tetra/Mixed 281
Robust (Octree) 281
Quick (Delaunay) 286
Smooth (Advancing Front) 287
Hexa-Dominant 288
Cartesian 288
Body-Fitted 289
Hexa-Core 293
Prism Meshing Parameters 294
Global Prism Settings 294
Prism Element Part Controls 305
Smoothing Options 306
Advanced Prism Meshing Parameters 307
Periodicity Set Up 312
Part Mesh Setup 315
Surface Mesh Setup 318
Curve Mesh Setup 320
Create Mesh Density 327
Define Connectors 329
Arbitrary Connectors 329
Bolt Weld Connectors 330
Seam Weld Connectors 330
Spot Weld Connectors 333
Spot Weld From File 335
Mesh Curve 336
Compute Mesh 336
Compute Surface Mesh 336
Compute Volume Mesh 337
Tetra/Mixed Mesh Type 337
Hexa-Dominant Mesh Type 341
Cartesian Mesh Type 341
Body-Fitted Mesh Method 341
Compute Prism Mesh 346
Blocking 349
Create Block 349
Initialize Block 350
From Vertices/Faces 357
3D Blocks 357
2D Blocks 362
Extrude Face 363
Interactive 363

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed Distance 363
Extrude Along Curve 364
2D to 3D Blocks 365
3D to 2D 371
Split Block 371
Split Block 372
Ogrid Block 373
Extend Split 376
Split Face 377
Split Vertices 378
Split Free Face 379
Imprint Free Face 379
Split Free Block 381
Merge Vertices 382
Merge Vertices 382
Merge Vertices by Tolerance 385
Collapse Block 385
Merge Vertex to Edge 385
Edit Block 388
Merge Blocks 388
Merge Faces 389
Modify Ogrid 389
Periodic Vertices 389
Convert Block Type 390
Change Block IJK 395
Renumber Blocks 395
Associate 395
Associate Vertex 396
Associate Edge to Curve 396
Associate Edge to Surface 397
Associate Face to Surface 398
Disassociate from Geometry 401
Update Associations 401
Reset Associations 403
Snap Project Vertices 404
Group/Ungroup Curves 405
Auto Associate 405
Move Vertex 405
Move Vertex 406
Set Location 407
Align Vertices 408
Align Vertices In-line 408
Set Edge Length 408
Move Face Vertices 409
Transform Blocks 409
Translate Blocks 410
Rotate Blocks 410
Mirror Blocks 410
Scale Blocks 411
Copy Periodic Blocking 411
Translate and Rotate 411
Edit Edge 412

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Split Edge 412
Unsplit Edge 414
Link Edge 414
Unlink Edge 414
Change Edge Split Type 415
Pre-Mesh Params 415
Update Sizes 415
Scale Sizes 418
Edge Params 419
Bunching Laws 422
Match Edges 425
Refinement 426
Pre-Mesh Quality 427
Pre Mesh Quality Options 428
Angle 429
Aspect Ratio 429
Constant Radius 429
Custom Quality 429
Determinant (2x2x2 stencil) 430
Determinant (3x3x3 stencil) 430
Distortion 430
Equiangle Skewness 430
Eriksson Skewness 431
Ford 431
Hex. Face Aspect Ratio 431
Hex. Face Distortion 431
Max Angle 431
Max Dihedral Angle 431
Max Length 431
Max Ortho 432
Max Ortho 4.3v 432
Max Ratio 432
Max Sector Volume 432
Max Side 432
Max Warp 432
Max Warp 4.3v 432
Mid Node 433
Mid Node Angle 433
Min Angle 433
Min Ortho 433
Min Sector Volume 433
Min Side 433
Opp Face Area Ratio 433
Opp. Face Parallelism 434
Orientation 434
Quality 434
Taper 435
Volume 435
Volume Change 435
Warpage 435
X Size 435
Y Size 435

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z Size 435
Pre-Mesh Quality Histogram 436
Pre-Mesh Smooth 437
Quality Method 437
Orthogonality Method 439
Multiblock Method 444
Block Checks 449
Delete Block 449
Edit Mesh 451
Create Elements 451
Node (Point Element) 452
Bar (Line Element) 453
Triangle 453
Quad 454
Tetra 455
Prism 455
Pyramid 456
Hexa 456
Auto Element Type 457
Extrude Mesh 458
Extrude by Element Normal 458
Extrude Along Curve 459
Extrude by Vector 460
Extrude by Rotation 461
Check Mesh 462
Errors 462
Possible Problems 463
Display Mesh Quality 465
Quality 466
Aspect Ratio 467
Aspect Ratio (Fluent) 469
Custom Quality 470
Determinant 470
Distortion 471
Element Stretch 471
Equiangle Skewness 471
Ford 471
Hex. Face Aspect Ratio 471
Hex. Face Distortion 472
Max Angle 472
Min Angle 472
Max Dihedral Angle 472
Max Length 472
Max Ortho 472
Min Ortho 472
Max Orthogls 472
Max Ratio 472
Max Sector Volume 472
Min Sector Volume 473
Max Side 473
Min Side 473
Min Side (Quad Optimized) 473

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Warp 473
Max Warpgls 473
Mesh Distribution 473
Mesh Expansion Factor 475
Mid Node 475
Mid Node Angle 475
Opp Face Area Ratio 475
Opp Face Parallelism 476
Orientation 476
Orthogonal Quality 476
Prism Thickness 478
Quadratic Dev 478
Skew 478
TGrid Skew 479
Surface Area 480
Surface Dev 480
Taper 480
Tetra Special 480
Volume 481
Volume Change 481
Volume/Area/Length 481
Workbench Shape 481
X Size 481
Y Size 482
Z Size 482
Quality Metric Histogram 482
Smooth Mesh Globally 483
Smooth Multiblock Domains Globally 488
Smooth Hexahedral Mesh Orthogonal 494
Repair Mesh 502
Build Mesh Topology 502
Remesh Elements 503
Remesh Bad Elements 505
Find/Close Holes in Mesh 506
Mesh From Edges 506
Stitch Edges 507
Smooth Surface Mesh 507
Flood Fill / Make Consistent 507
Associate Mesh With Geometry 510
Enforce Node, Remesh 510
Make/Remove Periodic 511
Mark Enclosed Elements 511
Merge Nodes 513
Merge Interactive 514
Merge Tolerance 516
Merge Meshes 516
Split Mesh 519
Split Nodes 520
Split Edges 521
Swap Edges 523
Split Tri Elements 525
Split Internal Wall 525

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y-Split Hexas at Vertex 525
Split Prisms 526
Move Nodes 527
Interactive 527
Exact 528
Offset Mesh 528
Align Nodes 529
Redistribute Prism Edge 529
Project Node to Surface 532
Project Node to Curve 533
Project Node to Point 533
Un-Project Nodes 534
Lock/Unlock Elements 534
Snap Project Nodes 534
Update Projection 535
Project Nodes to Plane 535
Transform Mesh 535
Translate 536
Rotate 537
Mirror 538
Scale 539
Translate and Rotate 539
Convert Mesh Type 540
Tri to Quad 541
Quad to Tri 541
Tetra to Hexa 542
All Types to Tetra 544
Shell to Solid 545
Create Mid Side Nodes 546
Delete Mid Side Nodes 549
Adjust Mesh Density 549
Refine All Mesh 550
Refine Selected Mesh 552
Coarsen All Mesh 552
Coarsen Selected Mesh 553
Renumber Mesh 554
User Defined 554
Optimize Bandwidth 555
Assign Mesh Thickness 555
Reorient Mesh 557
Reorient Volume 557
Reorient Consistent 557
Reverse Direction 558
Reorient Direction 558
Reverse Line Element Direction 558
Change Element IJK 558
Delete Nodes 558
Delete Elements 559
Edit Distributed Attribute 559
Properties 561
Load Material From File 562
Load Material From Library 562

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write Material File 563
Create Material Property 563
Isotropic 564
Shell Element Anisotropic 565
Solid Element Anisotropic 565
Shell Element Orthotropic 565
Isotropic Thermal Material 566
Anisotropic Thermal Material 566
Create Material Property Table 566
Define Point Element Properties 568
Define 1D Element Properties 570
Bar Element Properties 570
Rigid Elements Properties 571
Rod Connection Properties 572
Mass Connection Properties 572
Damper Connection Properties 572
Spring Properties 573
Viscous Damper Element Properties 573
Beam Element Properties 573
Gap Element Properties 574
Beam With Cross Section 575
Define 2D Element Properties 575
Shell Elements 576
Shear Elements 577
Layered Composite Elements 577
Define 3D Element Properties 578
Constraints 579
Create Constraint / Displacement 579
Create Constraint / Displacement on Point 581
Create Constraint / Displacement on Curve 581
Create Constraint / Displacement on Surface 582
Create Constraint / Displacement on Subset 583
Create Constraint / Displacement on Part 583
Create Constraint Equation 583
Define Constrained Node Sets 583
Define Contact 584
Automatic Detection 584
Manual Definition 586
Define Single Surface Contact 586
Define Initial Velocity 587
Define Planar Rigid Wall 587
Loads 589
Create Force 589
Create Force on Point 591
Create Force on Curve 591
Create Force on Surface 592
Create Force on Subset 592
Create Force on Part 592
Place Pressure 592
Place Pressure on Surface 593
Place Pressure on Subset 594
Place Pressure on Part 594

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

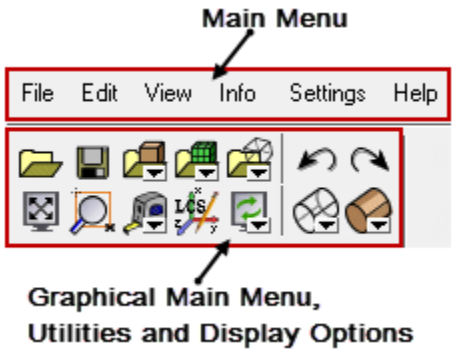
Create Temperature Boundary Condition 594
Temperature on Point 595
Temperature on Curve 595
Temperature on Surface 596
Temperature on Body 596
Temperature on Subset 597
Temperature on Part 597
Solve Options 599
Setup Solver Parameters 601
NASTRAN Setup Solver Parameters 602
ANSYS Setup Solver Parameters 607
LS-DYNA Setup Solver Parameters 608
Setup Analysis Type 609
NASTRAN Setup Analysis Type 610
Linear Static Analysis (Sol 101) 611
Modal (Sol 103) 612
Buckling Analysis (Sol 105) 612
Nonlinear Static (Sol 106) 612
Direct Frequency Response (Sol 109) 612
Direct Transient Response (Sol 109) 613
Modal Frequency Response (Sol 111) 613
ANSYS Setup Analysis Type 614
Structural Analysis 614
Thermal 615
LS-DYNA Setup Analysis Type 616
Abaqus Setup Analysis Type 617
Autodyn Setup Analysis Type 618
Setup a Subcase 618
NASTRAN Setup a Subcase 619
ANSYS Setup a Subcase 620
Abaqus Setup a Subcase 621
Write/View Input File 622
NASTRAN Write/View Input File 622
ANSYS Write/View Input File 623
LS-DYNA Write/View Input File 625
Abaqus Write/View Input File 626
Autodyn Write/View Input File 627
Submit Solver Run 628
NASTRAN Submit Solver Run 628
ANSYS Submit Solver Run 629
LS-DYNA Submit Solver Run 630
Output 631
Select Solver 631
ANSYS 633
ANSYS CFX 646
ANSYS Fluent 647
CGNS 649
Polyflow 651
Boundary Conditions 653
Edit Parameters 655
Write Input 655
ANSYS Input Files 655

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu Area

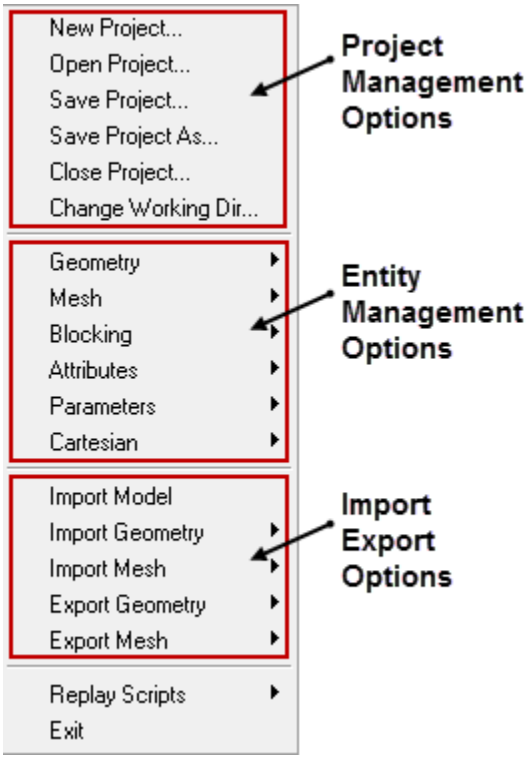
The Main Menu area contains drop down menus for managing projects, files, and settings. It also includes Utility icons for common tasks and display options.

Figure 1: Main Menu Area



File Menu

Figure 2: File menu



The File menu commands for managing your project, for managing the entities that comprise a project, for importing and exporting data, for managing Replay Scripts, and to quit the program.

Project Management Options

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity Management Options
Import/Export Options
Replay Scripts
Exit

Project Management Options

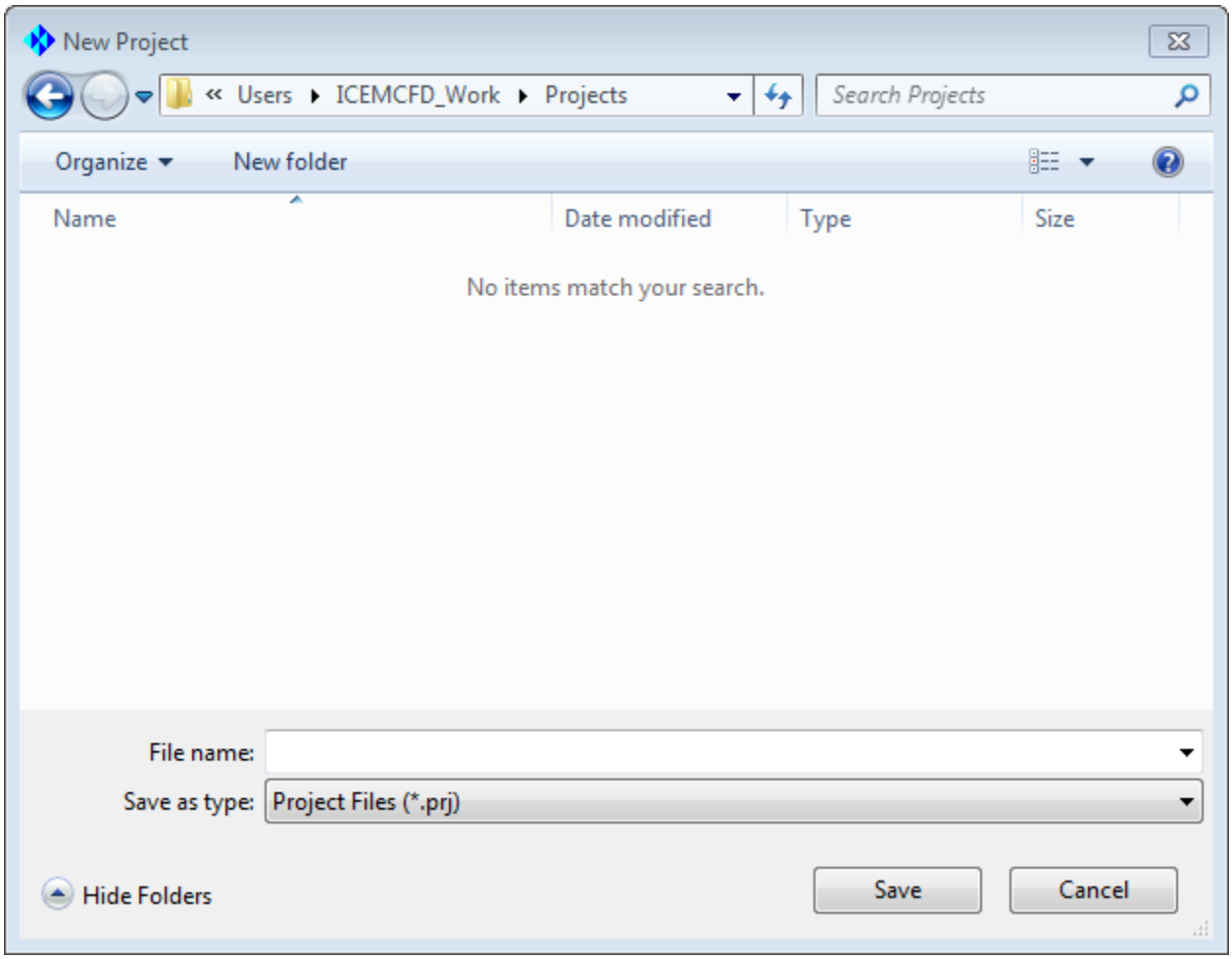
Your ANSYS ICEM CFD work is organized into Project files (`*.prj`). A project file contains the information necessary to manage several data files associated with your project. The data files in a project are a Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Attributes files (`*.fbc` and `*.atr`), and Parameters file (`*.par`).

Options for managing your project file include the following:

New Project

To initiate a project file, select the **New Project** option.

Figure 3: New Project Window



- Browse to the desired folder. This location becomes the working folder for all files associated with your project.
- You must specify a **File name**.
- The only available **Save as type** option is .prj for project files.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Save**.

Note

- Existing project files in the current folder will be displayed.
- You may choose a **File name** from the existing files displayed or from a list of the most recently opened project files using the drop down list, although a new project name is advised to prevent overwriting an existing project.
- If you already have a project open, this option will close the current project.

Open Project

To work on an existing project, select the **Open Project** option.

Navigate to the working folder and select the desired project from the **Open Project** window; or select an existing project from the drop down list in the **File name** field.

Note

ANSYS ICEM CFD supports native projects (`.prj` extension) or ANSYS Workbench projects (`.wbpj` extension).

Save Project

To update the project file (`*.prj`) on your disk, select the **Save Project** option.

If you are working in an existing project, this option will overwrite that project. If not, you will be prompted for a **File name** and location.

This option also updates and saves the data files associated with the project. The files saved are the Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Parameters file (`*.par`) and Attributes files (`*.fbc` and `*.atr`).

Save Project As

To create a new project file with existing work, select the **Save Project As** option.

The **Save Project As** dialog box opens, prompting you for a **File name** and location for your project.

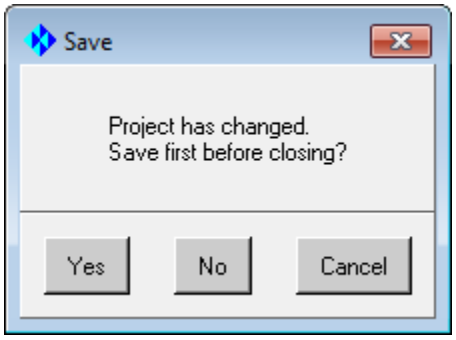
Close Project

To unload all project data and leave your ICEM CFD session open, select the **Close Project** option.

If the project has changed, you will be prompted to save your changes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4: Save Confirmation Window

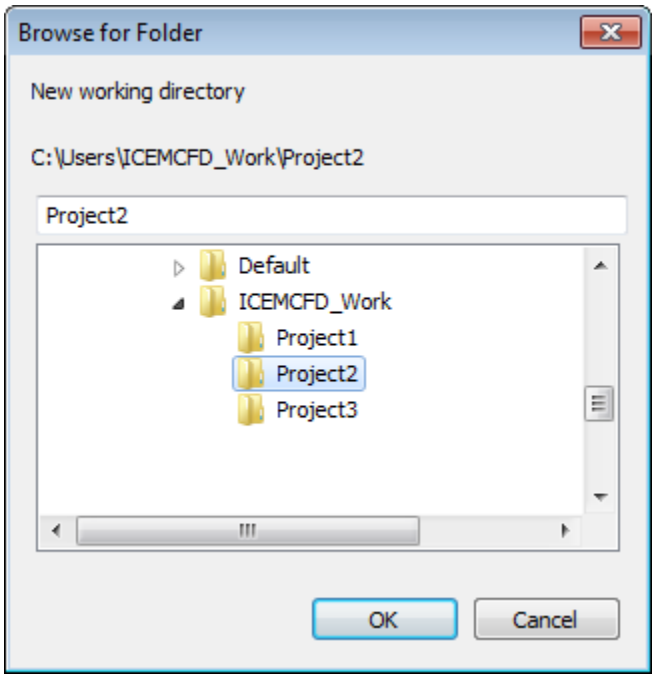


Change Working Directory

To specify a default directory where the project and data files will be saved, select the **Change Working Directory** option.

You can directly enter the path name or browse to the desired working directory for the project.

Figure 5: Change Working Directory Window



Entity Management Options

Your ANSYS ICEM CFD project consists of several entities, each with its own data which may be managed independently. Each option in this section opens a submenu to manage that entity's data.

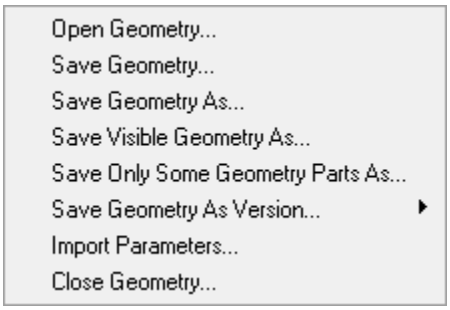
Geometry
Mesh
Blocking
Attributes
Parameters
Cartesian

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry

The geometry file (`*.tin`) includes details of the shapes, names, material properties, and so on of all component parts in the project. Options for managing the geometry file are shown in Figure 6: Geometry Options (p. 5).

Figure 6: Geometry Options



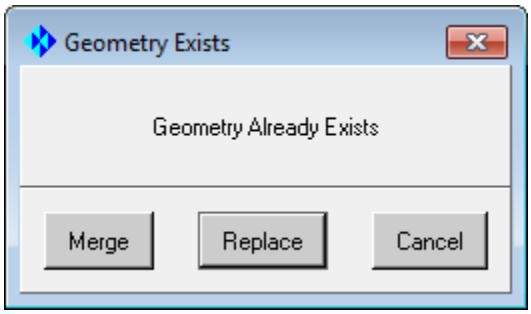
Open Geometry

To load a geometry file into memory and display its graphical image, select the **Geometry > Open Geometry** option.

Select the required geometry file (`*.tin`) in the **Open** window. You may have to browse to a different working directory.

If a geometry is already loaded, the **Geometry Exists** window will open, prompting you to either merge the new geometry with the existing geometry or replace the existing geometry.

Figure 7: Geometry Exists Window



Save Geometry

To update the `.tin` file with all your recent geometry work, select the **Geometry > Save Geometry** option.

This option will overwrite an existing, open geometry file. If no geometry file is open, a **Save all geometry** window will prompt you for a File name and location.

Save Geometry As

To create a new `.tin` file with existing geometry data, select the **Geometry > Save Geometry As** option.

The **Save all geometry** window opens where you are prompted for a File name and location.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save Visible Geometry As

To create a new `.tin` file containing only those geometry entities currently displayed, select the **Geometry > Save Visible Geometry As** option. This is typically done when you need to work with a subset of all geometry in your project.

The **Save only visible geometry** window opens where you are prompted for a File name and location.

Save Only Some Geometry Parts As

To create a new `.tin` file containing a subset of all geometry in your project, select the **Geometry > Save Only Some Geometry Parts As** option.

The **Save only some parts** window opens where you are prompted for a File name and location, followed by the **Select parts** dialog box for you to choose which parts are saved in the new file.

Save Geometry As Version ...

To save the geometry in an earlier ANSYS ICEM CFD format, select the **Geometry > Save Geometry As Version** option. Options available are Version 4.3, Version 10.0, or Version 13.0.

Note

- Options related to Mesh Type and Mesh Method introduced in ANSYS ICEM CFD 11.0 will be ignored when saving a geometry file in the ANSYS ICEM CFD 10.0 or earlier format.
- The parameter related to curves and surfaces introduced in ANSYS ICEM CFD 14.0 will be ignored when saving the geometry file in the ANSYS ICEM CFD 13.0 or earlier format.

Note

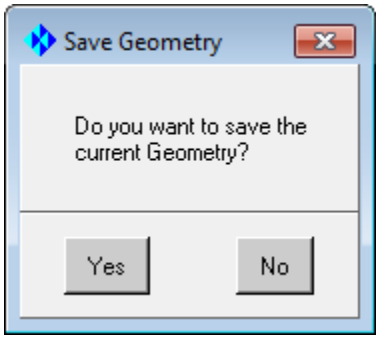
This parameter is introduced because of changes to AutoVT in ANSYS Workbench and can be introduced when the model comes through ANSYS Workbench, including with the Workbench CAD interfaces.

Close Geometry

To remove the geometry information from the graphical display and close the loaded geometry file, select the **Geometry > Close Geometry** option.

If the geometry has been modified, the **Save Geometry** window will open, asking if you want to save the modified geometry.

Figure 8: Save Geometry Window



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh

The mesh file (*.uns) includes details of the line, shell, and volume mesh elements of the project. Options for managing the mesh file are shown in Figure 9: Mesh Options (p. 7).

Figure 9: Mesh Options

Open Mesh

To load a mesh file into your project and display its image, select the **Mesh > Open Mesh** option.

Select the required mesh file (*.uns) in the **Open** dialog box. You may have to browse to a different working directory.

If a mesh file is already loaded in the project, the **Mesh Exists** window will open, prompting you to either merge the new mesh with the existing mesh or replace the existing mesh (similar to Figure 7: Geometry Exists Window (p. 5)).

Open Mesh Shells Only

To load only a shell (surface) mesh, select the **Mesh > Open Mesh Shells Only** option.

Select the required mesh file in the **Open** window. If a mesh file is already loaded in the project, the **Mesh Exists** window will open, prompting you to either merge with, or replace, the existing mesh.

Load from Blocking

To load the mesh from the currently loaded blocking file, select the **Mesh > Load from Blocking** option.

The existing mesh will be converted from structured to unstructured mesh; and the meshing entities in the display tree will be updated.

Save Mesh

To update your .uns file with all your recent mesh work, select the **Mesh > Save Mesh** option.

This option will overwrite an existing, open mesh file. If no mesh file is open, a **Save all mesh** window will prompt you for a File name and location.

Save Mesh As

To create a new .uns file with existing mesh data, select the **Mesh > Save Mesh As** option,

The **Save all mesh** window opens where you will be prompted for a File name and location.

Save Visible Mesh As

To create a new .uns file containing only those mesh entities currently displayed, select the **Mesh > Save Visible Mesh As** option. This is typically done when you need to work with a subset of all mesh in your project.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

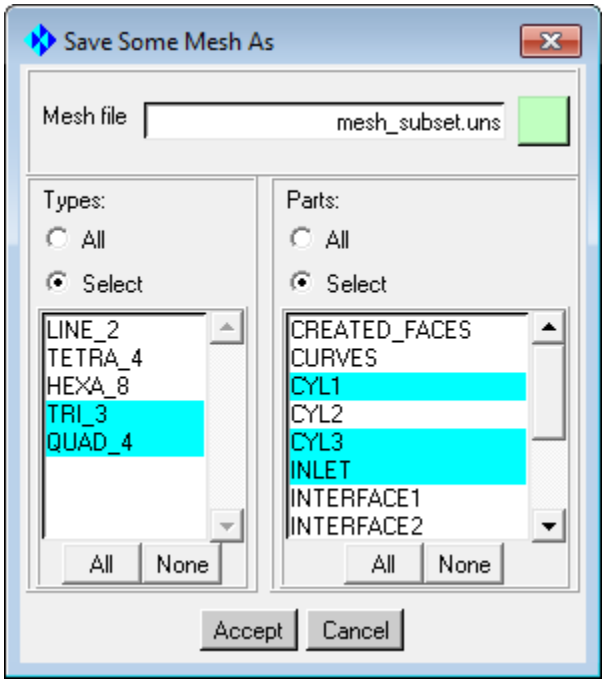
The **Save only visible mesh** window opens where you will be prompted for a File name and location.

Save Only Some Mesh As

To create a new .uns file containing a subset of your mesh data, select the **Mesh > Save Only Some Mesh As** option.

The **Save Some Mesh As** dialog box opens where you select the mesh types and/or mesh parts to be saved in the new file. See Figure 10: Save Some Mesh As Dialog (p. 8).

Figure 10: Save Some Mesh As Dialog



Mesh file

The currently loaded **Mesh file** name will appear here. You may use the existing file name, type a new Mesh file name, or browse for a file by clicking on the box to the right of the field.

Types

Select **All** to include all mesh types, or **Select** to choose the mesh elements to be saved. The selected types will be highlighted.

Parts

Select **All** to include the mesh associated with all the existing parts, or **Select** to choose the mesh parts to be saved. The selected parts will be highlighted.

Close Mesh

To remove the mesh information from the graphical display and close the loaded domain file or mesh file, select the **Mesh > Close Mesh** option.

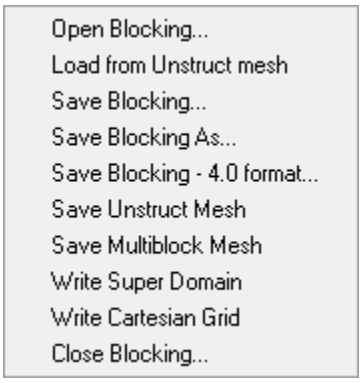
If the mesh has been modified, the **Save Mesh** window will open, asking if you want to save the modified mesh (similar to Figure 8: Save Geometry Window (p. 6)).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blocking

The blocking file (`*.blk`) includes details of the underlying framework used to create a structured hexahedral mesh in your project. Blocking files may also be loaded from or saved to unstructured mesh. The options for managing the blocking file are shown in Figure 11: Blocking Options (p. 9).

Figure 11: Blocking Options



Open Blocking

To load a blocking file into your project and display its image, use the **Blocking > Open Blocking** option.

Select the desired blocking file from the **Open** dialog box. You may have to browse to different working directory.

If a blocking file is already loaded in the project, the **Blocking Exists** window will open, prompting you to either merge the new blocking with the existing blocking or replace the existing blocking (similar to Figure 7: Geometry Exists Window (p. 5)).

Load from Unstruct mesh

To create a hexa blocking file from an unstructured mesh composed entirely of hexa elements, select the **Blocking > Load from Unstruct mesh** option. This option is useful to regenerate a blocking file from a hexa mesh that was originally created from a blocking file, and is most robust when used in this fashion. This option can also be used on unstructured meshes originally written from external Multiblock formats such as Plot3D.

It is also useful to create complicated 2D blocking by first creating 2D elements. Each quad element becomes a 2D hexa block. Complicated block structures can be made this way, which may not be possible to do in a top-down method.

Save Blocking

To update your `.blk` file with all your recent blocking work, select the **Blocking > Save Blocking** option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If an existing blocking file, `filename.blk` is already open, then this option will rename the existing blocking file to `filename.blk0` (and then sequentially `filename.blk1`, `filename.blk2`, etc.) and save the current file to `filename.blk`.

Note

The automatic indexing of blocking file names makes it easy to jump back to an earlier step if you wish.

Note

A key strength of ANSYS ICEM CFD Hexa is the separate blocking layer file. This blocking file sit on top of the geometry (`*.tin` file) and is associated with it, but can be loaded with other topologically similar models and associated with them. In this way a blocking structure can be used with a family of geometries to rapidly create a series of high quality meshes or shared between colleagues working on similar tasks.

Save Blocking As

To create a new `.blk` file with existing blocking data, select the **Blocking > Save Blocking As** option.

The **Save as** window opens where you will be prompted for a File name and location.

Save Blocking as 4.0 Format

To save the blocking file in the legacy ANSYS ICEM CFD 4.0 format, select the **Blocking > Save Blocking – 4.0 format** option.

Save Unstruct Mesh

To save the existing blocking pre-mesh in an unstructured hexa mesh format (`*.uns` file), select the **Blocking > Save Unstruct Mesh** option.

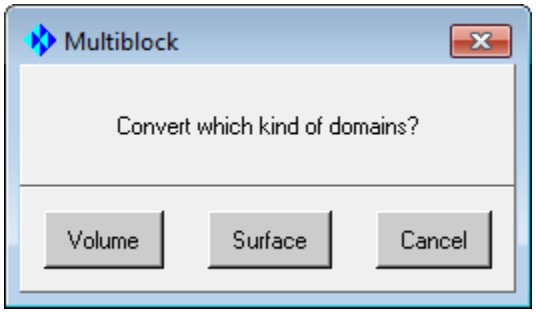
The **Save Mesh as** dialog box opens where you will be prompted for a File name and location. The unstructured mesh is not loaded.

Save Multiblock Mesh

To save the current blocking data in the Multiblock structured format, select the **Blocking > Save Multiblock Mesh** option.

A dialog box will open prompting you to choose which type of domains to convert.

Figure 12: Convert Multiblock



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write Super Domain

To save the current blocking data in a merged, one-domain Multiblock mesh, select the **Blocking > Write Super Domain** option. The intended solver must be compatible with the Super Domain format.

Write Cartesian Grid

To save the current blocking data in a Cartesian grid format, select the **Blocking > Write Cartesian Grid** option. This is useful as a starting point for the Body-Fitted Cartesian (BFCart) meshing module (see the **Cartesian file** information on the Body-Fitted Mesh Method page).

To use this option, start by creating a block from the geometry extents. You can modify the vertex locations, but ensure the block remains aligned with a Cartesian coordinate system (right angled corners). Control the Cartesian grid distribution and biasing with splits and edge parameters. Also, edge distributions should be "copied to parallel" to maintain a consistent Cartesian distribution throughout the grid.

Close Blocking

To remove the blocking information from the graphical display and close the loaded blocking file, select the **Blocking > Close Blocking** option.

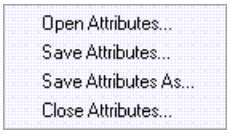
If the blocking has been modified, the **Save Blocking** window will open, asking if you want to save the modified blocking (similar to Figure 8: Save Geometry Window (p. 6)).

Attributes

The attributes files (extension `*.fbc` or `*.atr`) maintain the association of user-specified data for parts, element properties, loads and constraints with nodes/elements of the mesh for a project. The files are updated automatically to preserve consistency with the mesh, every time the project is saved.

You can perform the following operations on an existing attributes file as shown in Figure 13: Attributes Options (p. 11).

Figure 13: Attributes Options



Open Attributes

To load an attributes file (`*.atr` or `*.fbc`), select the **Attributes > Open Attributes** option.

Select the desired attributes file from the **Open** dialog box, or select an existing file from the drop down list in the **File name** field. You may have to browse to the working folder.

Save Attributes

To update your `.fbc` and `.atr` files with all your recent boundary conditions and attributes work, select the **Attributes > Save Attributes** option.

This option will overwrite existing, open attributes files. If no attribute file is open, a **Save as** window will prompt you for a File name and location.

Save Attributes As

To create new `.fbc` and `.atr` files with existing attributes data, select the **Attributes > Save Attributes As** option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Save as** window opens where you will be prompted for a File name and location.

Close Attributes

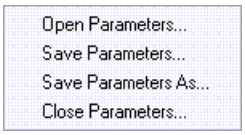
To close the attributes files, select the **Attributes > Close Attributes** option.

Parameters

The parameters file (extension `*.par`) contains mesh-independent data such as material properties, local coordinate systems, solver analysis set up and run parameters. The data in the parameters file is cross-referred in the attributes file when a set of parameters is associated to the nodes/elements of the mesh. The parameters file is also updated automatically along with the attributes files whenever a project is saved.

You can perform the following operations on an existing parameters file as shown in Figure 14: Parameters Options (p. 12).

Figure 14: Parameters Options



Open Parameters

To load a parameters file (`*.par`), select the **Parameters > Open Parameters** option.

Select the desired file from the **Open** dialog box; or select an existing file from the drop down list in the **File name** field. You may have to browse to the working folder.

Save Parameters

To update your `.par` file with all your recent parameters work, select the **Parameters > Save Parameters** option.

This option will overwrite an existing, open parameters file. If no parameter file is open, a **Save as** window will prompt you for a File name and location.

Save Parameters As

To create new `.par` file with existing parameters data, select the **Parameters > Save Parameters As** option.

The **Save as** window opens where you will be prompted for a File name and location.

Close Parameters

To close the parameters file, select the **Parameters > Close Parameters** option.

Note

The data format for both the attributes and parameters files is unique for a solver. The default format is Nastran. The conversion of Nastran files to ANSYS & LS-DYNA files is done automatically depending on the selected solver. The other solver-specific attributes and parameters can be specified through the **Edit Parameters** and/or **Edit Attributes** options available under **Advanced Edit Options** in the **Write/View Input File** DEZ. After selecting this option, the **Solver Parameters** window will open and you can set the required parameters.

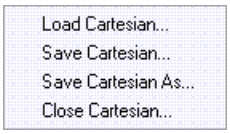
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian

The Cartesian file (extension `*.crt`) contains information regarding the Cartesian grid for your project, if one has been created. See **Write Cartesian Grid** under Blocking (p. 9).

You can perform the following operations on an existing Cartesian file as shown in Figure 15: Cartesian Options (p. 13).

Figure 15: Cartesian Options



Load Cartesian

To load a Cartesian grid file (`*.crt`), select the **Cartesian > Load Cartesian** option.

Select the desired file from the **Open** dialog box, or select an existing file from the drop down list in the **File name** field. You may have to browse to the working folder.

Save Cartesian

To update your `.crt` file with all your recent Cartesian grid work, select the **Cartesian > Save Cartesian** option.

This option will overwrite an existing, open Cartesian file. If no Cartesian file is open, a **Save as** window will prompt you for a File name and location.

Save Cartesian As

To create a new `.crt` file with existing Cartesian grid data, select the **Cartesian > Save Cartesian As** option.

The **Save as** window opens where you will be prompted for a File name and location.

Close Cartesian

To close the Cartesian file, select the **Cartesian > Close Cartesian** option.

Import/Export Options

ANSYS ICEM CFD is capable of importing or exporting Geometry and parts data from/to many popular CAD software programs, as well as importing or exporting Mesh data from/to many third party Solvers. The **Import Model** option uses available **Workbench Readers** to extend the options for interfacing with third party software.

Import Model
Import Geometry
Import Mesh
Export Geometry
Export Mesh

Note

The special characters Ä, ä, Ç, Ö, ö, Ü, and ü are not supported in file names on Linux.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Model

The **Import Model** option allows you to use the Workbench Readers to import many kinds of files into ANSYS ICEM CFD.

The following table lists the ANSYS product file types that may be imported using this option.

ANSYS licensed product file types that may be imported using Workbench Readers.	
ANSYS	*.agdb, *.anf, *.cmdb, *.dsdb, *.mechdat, *.meshdat, *.meshdb, *.mshdat, *.mshdb, *.wbpj
BladeGen	*.bgd
DesignModeler	*.agdb
GAMBIT	*.dbs
SpaceClaim	*.scdoc

Table 1: Geometry Interface Support using Workbench Readers (p. 14) lists the third party CAD software formats supported by the **Import Model** option (√ = supported).

Table 1: Geometry Interface Support using Workbench Readers

		Windows 32 (Intel IA32)		**Windows x64 (EM64T, AMD64)**			**Linux x64 (Intel Xeon EM64T, AMD64)**		
CAD Product (file types)	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8 (Pro & Enterprise)**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
Reader for ACIS (*.sat, *.sab)	24	√	√	√	√	√	√	√	√
Reader for AutoCAD (*.dwg, *.dxf)	2013	√	√	√	√	√			
	2014		√		√	√			
Reader for CATIA V4 (*.model, *.dlv)	4.2.4	√	√	√	√	√			
Reader for CATIA V5 (*.CATPart, *.CATProduct)	V5-6R2013	√	√	√	√	√			

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		Windows 32 (Intel IA32)		Windows x64 (EM64T, AMD64)			Linux x64 (Intel Xeon EM64T, AMD64)		
CAD Product (file types)	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8 (Pro & Enterprise)**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
Reader for CATIA V5 - CADNexus CAPRI CAE Gateway V3.16.1 (*.CATPart, *.CATProduct)	V5R21, V5–6R2012, V5–6R2013	√	√	√	√				
Reader for CATIA V6 (*.CATPart, *.CATProduct)	R2013	√	√	√	√	√			
Creo Elements/Direct Modeling (*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc)	17.0	√	√	√	√				
	18.0	√	√	√	√				
	18.1	√	√	√	√	√ (requires Creo Elements/Direct Modeling M030 or later)			
Reader/Plugin for Creo Elements/Pro (Pro/ENGINEER) (*.prt, *.asm, *.prt*, *.asm*)	Wildfire 5	√	√ (requires Pro/E Wildfire 5 M020)	√	√ (requires Pro/E Wildfire 5 M020)				
	Creo Parametric 1.0	√	√	√	√				
	Creo Parametric 2.0	√	√	√	√	√ (requires Creo Para-			

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		Windows 32 (Intel IA32)		Windows x64 (EM64T, AMD64)			Linux x64 (Intel Xeon EM64T, AMD64)		
CAD Product (file types)	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8 (Pro & Enterprise)**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
						metric 2.0 M030)			
Reader for Creo Elements/Pro (Pro/ENGINEER) (*.prt, *.asm, *.prt*, *.asm*)	Creo Parametric 2.0	√	√	√	√	√			
Reader/Plugin for Autodesk Inventor (*.iam, *.ipt)	2013	√	√	√	√	√ (requires Autodesk Inventor 2013 SP2)			
	2014		√		√	√			
Reader for Autodesk Inventor (*.iam, *.ipt)	2013	√	√	√	√	√			
Reader for JT (*.jt)	9.5	√	√	√	√	√			
Reader for Parasolid (*.x_t, *x_b, *.x_txt, *.xmt_b, *.xmt_bin)	26.0	√	√	√	√	√	√	√	√
Reader for Solid Edge (*.asm, *.par,	ST5		√		√				
	ST6		√		√	√			

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		Windows 32 (Intel IA32)		Windows x64 (EM64T, AMD64)			Linux x64 (Intel Xeon EM64T, AMD64)		
CAD Product (file types)	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8 (Pro & Enterprise)**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
*.psm, *.pwd)									
Reader for SolidWorks (*.sldasm, *.sldprt)	2013	√	√	√	√	√			
Reader/Plugin for SolidWorks (*.sldasm, *.sldprt)	2012	√	√	√	√				
	2013		√		√	√			
Reader for STEP (*.stp, *.step)	AP203, AP214	√	√	√	√	√	√	√	√
Reader for STL (*.stl)		√	√	√	√	√	√	√	√
Reader for NX (*.prt)	8.5	√	√	√	√	√			
Reader/Plugin for NX (*.prt)	8.5		√		√	√	√		√
	8.0	√	√	√	√		√		√
	7.5	√	√	√	√			√	
Plugin for TeamCenter Engineering	Unified Architecture 8.1.0.4 (2-tier only) with NX7.5	√		√					
	Unified Architecture 8.3 (2- and 4-tier) with NX7.5.4 MP2 or higher, NX8.0 (8.3.3.4 or	√		√					

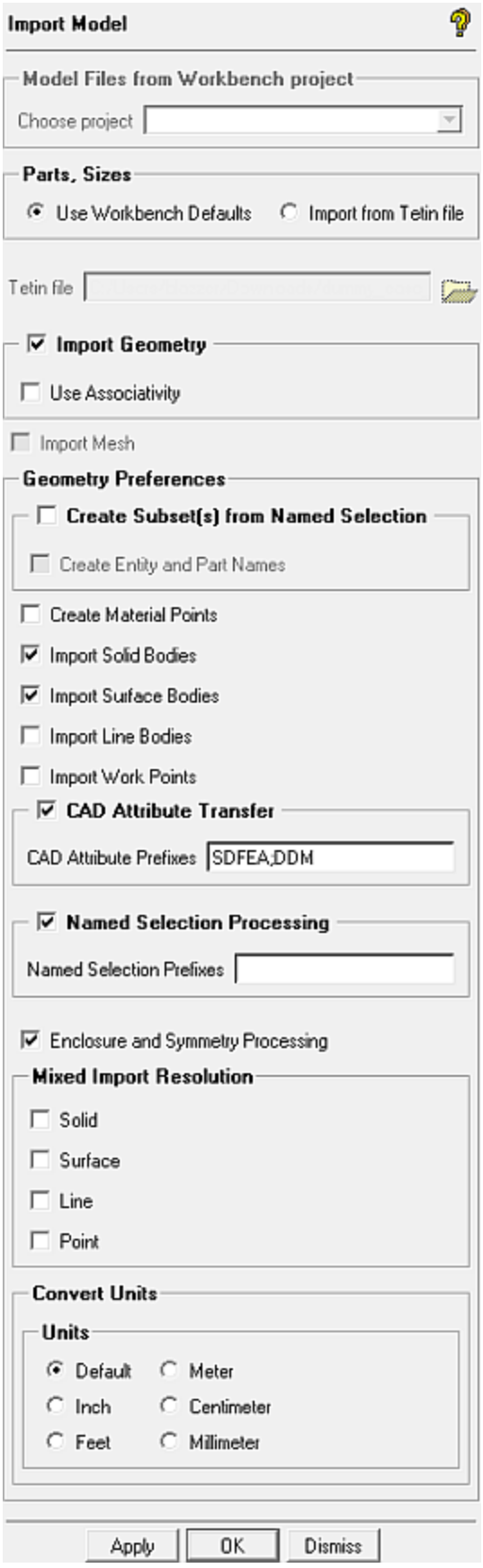
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		Windows 32 (Intel IA32)		Windows x64 (EM64T, AMD64)			Linux x64 (Intel Xeon EM64T, AMD64)		
CAD Product (file types)	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8 (Pro & Enterprise)**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
	higher) & NX8.5 (8.3.3.5 or higher)								
	Unified Architecture 9.1 (2- & 4-tier) with NX7.5, NX8.0 & NX8.5	√		√		√ (requires patch 9.1.2.3)			

Note

This option only appears if ANSYS Workbench is installed along side ANSYS ICEM CFD.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Files from Workbench project

When the Workbench project (*.wbpj) file is read using the **Import Model** option, the existing geometry and/or mesh files will be located. You can then select the specific data file to be imported.

Selecting at the project level is more convenient than locating the particular Workbench geometry or mesh files within the directory structure. However, the **Model Files from Workbench project** option is particularly important if there are multiple files located.

Parts, Sizes

allows you to replace the default Workbench parameters with the parameters defined in an existing tetin file to the imported geometry. The selected tetin file would transfer its global, part, and entity based settings to the newly imported geometry. This results in rapid meshing parameter setup for a modified geometry.

Import Geometry

allows you to import geometry. In general, the geometry import options available follow the options available in ANSYS Workbench Meshing. All the options under **Geometry Preferences** will be active only when **Import Geometry** is enabled.

Use Associativity

Default is OFF. If checked ON, ICEM CFD will use a default part manager database to store persistency data.

Note

- Valid extensions are .prt .asm .prt* .asm* .par .psm .pwd .CATPart .CATProduct .sldasm .sldprt .ipt and .iam. Otherwise you will get an error message about an invalid file format.
- It is required that your ICEM CFD project exist to store the associated geometry file.

Import Mesh

allows you to import meshes from *.meshdat. or *.mechdat files. Legacy formats such as *.cmdb and *.dsdb are also supported. When only **Import Mesh** is enabled, all the **Geometry Preferences** options will be deactivated.

Geometry Preferences

allows you to set the following geometry preferences:

Create Subset(s) from Named Selection

enables the creation of geometry subsets instead of parts, thereby allowing you to decide which part the geometry entity (point/curve/surface) should be associated to. When this option is disabled each geometry entity (point/curve/surface) will be assigned to only one part, thereby losing association with other Named Selections (which are mapped to parts). This option is available only when the **Import Mesh** option is disabled.

Create Entity and Part Names

uses attribute/entity names assigned in the CAD file to create entity and part names in ICEM CFD.

Create Material Points

enables the automatic creation of material points during import.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Solid Bodies
enables the import of solid bodies.

Import Surface Bodies
enables the import of surface bodies.

Import Line Bodies
enables the import of line bodies.

Import Work Points
enables the import of work points.

CAD Attribute Transfer
allows import of CAD system attributes into ANSYS ICEM CFD.

CAD Attribute Prefixes
allows you to set the CAD attribute prefix when **CAD Attribute Transfer** is enabled. By default the filter is set to **SDFEA;DDM**. If the filter is set to an empty string, all applicable entities will be imported as CAD system attributes. You may enter multiple prefixes with each prefix delimited by a semicolon.

Named Selection Processing
creates a named selection based on data generated in the CAD system.

Named Selection Prefixes
allows you to set the named selection processing prefix key when **Named Selection Processing** is enabled. If the filter is set to an empty string, all applicable entities will be imported as named selections. You may enter multiple prefixes with each prefix delimited by a semicolon.

Enclosure and Symmetry Processing
enables the processing of enclosure and symmetry named selections. This option is enabled by default.

Mixed Import Resolution
allows parts of mixed dimension to be imported as components of assemblies which have parts of different dimension. By default, no bodies from a multibody part will be transferred to ANSYS ICEM CFD. You can select the appropriate combination of bodies to be transferred to ANSYS ICEM CFD using the following options:

Solid
solid(s) from the selected geometry's multibody part(s) are imported into ANSYS ICEM CFD.

Surface
surface(s) from the selected geometry's multibody part(s) are imported into ANSYS ICEM CFD.

Line
line(s) from the selected geometry's multibody part(s) are imported into ANSYS ICEM CFD.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Point
point(s) from the selected geometry's multibody part(s) are imported into ANSYS ICEM CFD.

Note

The **Mixed Import Resolution** options are valid only for "real" CAD files (Parasolid, UniGraphics, CATIA, etc.). For ANSYS Workbench (*.agdb) files, only the primary options (**Import Solid Bodies**, **Import Surface Bodies**, **Import Line Bodies**) are valid.

Convert Units
allows you to scale the imported geometry/mesh based on the units specified.

Default
retains the original units from the imported file.

Inch
scales the imported geometry/mesh to inches (in).

Feet
scales the imported geometry/mesh to feet (ft).

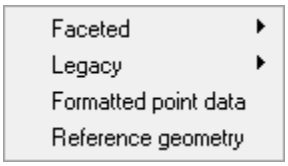
Meter
scales the imported geometry/mesh to meters (m).

Centimeter
scales the imported geometry/mesh to centimeters (cm).

Millimeter
scales the imported geometry/mesh to millimeters (mm).

Import Geometry

Figure 16: Import Geometry Options



The **Import Geometry** option allows you to import geometry from various sources, including several faceted geometry formats (Nastran, Patran, STL, or VRML) or CAD drawing packages (CATIA, DWG, Parasolid, Rhino 3DM, STEP/IGES, etc.). When you select the CAD software format of the geometry file you want to import; a corresponding window will pop up with further options.

Options for importing geometry are shown in Figure 16: Import Geometry Options (p. 22).

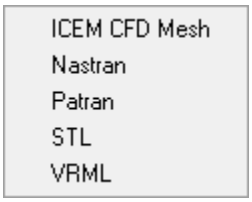
For detailed CAD-related information specific to ANSYS Workbench, see the CAD Integration section of the product help. Within the **Help Viewer**, expand the **CAD Integration** section in the hierarchical tree.
Faceted
Legacy
Formatted Point Data
Reference Geometry

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faceted

The **Faceted** submenu gives several options for importing faceted geometry.

Figure 17: Import Faceted Geometry



Options for importing faceted geometry are shown in Figure 17: Import Faceted Geometry (p. 23).

ICEM CFD Mesh
Nastran
Patran
STL
VRML

ICEM CFD Mesh

The **ICEM CFD Mesh** option allows you to import unstructured mesh file as triangulated surface data. The volume elements in the mesh file will be ignored. Surfaces and curves are segmented by part.

Note

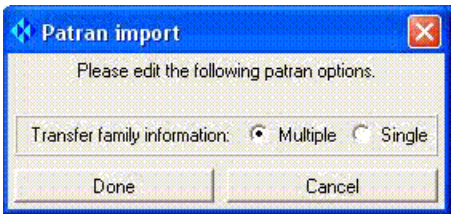
You can convert loaded mesh to facets using the Mesh>Facets option (**Edit > Mesh>Facets**).

Nastran

The **Nastran** option allows you to import a Nastran surface mesh file (*.dat, *.bdf) as triangulated surface data.

Patran

The **Patran** option allows you to import Patran files (*.pat).



Transfer family information

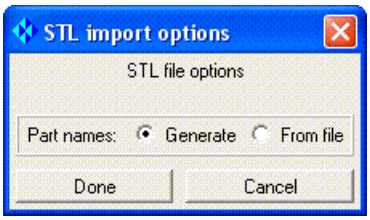
Multiple
converts each Patran group (PID) into an ICEM CFD part.

Single
brings the Patran groups all in as one part.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STL

The **STL** option allows you to import a Stereolithography Part file (*.stl) as triangulated surface data. Note that the parts defined in the STL file will also be imported. Select the format for the part names and click **Done** to finish the operation.



Part names

Generate
generates part names of the format **`Filename.PartName`**, where **`PartName`** is the part name defined in the STL file.

From file
retains the part names as defined in the STL file.

Note

This option is valid only for STL files in ASCII format. For STL files in binary format, part names of the format **`Filename.PartName`** will be generated as described.

VRML

The **VRML** option allows you to import a VRML file as triangulated surface data (*.wrl).

Note

You can only import VRML files in ASCII format. The import of binary VRML files in not supported in **ANSYS ICEM CFD**.

Legacy

The **Legacy** option allows you to import geometry from various CAD formats such as CATIA V4, Rhino 3D, IGES, and Parasolid, etc. Select the format of the geometry file you want to import; a corresponding window will pop up with further options.

Note

Some higher formats are supported in the **Import Model** option using Workbench Readers. If your model fails to read with the ANSYS ICEM CFD **Legacy** option, try the **Import Model** option instead.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The supported legacy geometry interfaces are listed in Table 2: Legacy Geometry Interface Support (p. 25) (√ = supported).

Table 2: Legacy Geometry Interface Support

		Windows 32 (Intel IA32)		Windows x64 (EM64T, AMD64)			Linux x64 (Intel Xeon EM64T, AMD64)		
CAD Product	**Version of CAD Package**	**MS Windows XP**	**MS Windows 7**	**MS Windows XP**	**MS Windows 7**	**MS Windows 8**	**Red Hat 6**	**Red Hat 5**	**SuSE ES 11**
Reader for ACIS	18.0.1	√	√	√	√	√	√	√	√
Reader for CATIA V4	4.2.4	√	√	√	√	√	√	√	√
Reader for DWG		√	√	√	√	√	√	√	√
Reader for GEMS		√	√	√	√	√	√	√	√
Reader for IDI	MS 8/9	√	√	√	√	√			
Reader for Parasolid	24.0	√	√	√	√	√	√	√	√
Reader for Rhino 3DM		√	√	√	√	√			
Reader for Plot3d		√	√	√	√	√			
Reader for IGES	4.0, 5.2, 5.3	√	√	√	√	√	√	√	√
Reader for STEP	AP203, AP214	√	√	√	√	√	√	√	√

Figure 18: Import Legacy Geometry

Options for importing geometry are shown in Figure 18: Import Legacy Geometry (p. 25).

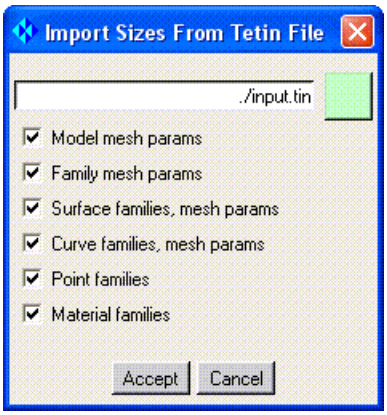
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acis
CATIA V4
DWG
GEMS
IDI
ParaSolid
Rhino 3DM
Plot3d
STEP / IGES

Acis

The **Acis** option converts an Acis file (*.sat) to a Geometry file (*.tin) and loads it. Select an Acis file in the file browser window, and click **Open**. The **Import Sizes From Tetin File** window will open where you can select a previously setup version of the model and transfer parts and mesh parameters to the newly imported geometry. This saves time that would have been spent setting up the new model.

Figure 19: Import Acis Geometry window



Model mesh params
are the global mesh settings.

Surface families, mesh params
will put the imported surface entities into parts (same as the original) and assign surface based mesh parameters.

Curve families, mesh params
will put the imported curve entities into parts (same as the original) and assign curve based mesh parameters.

Point families
will put the imported point entities into parts (same as the original). Point entities do not support mesh parameters.

Note

These surface/curve/point entity options are based on the entity names matching between the source file and the imported file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CATIA V4

The **CATIA V4** option converts a CATIA file (*.model) to a Geometry file (*.tin) and loads it. Select a CATIA file in the file browser window, and click **Open**. This will open the **Import Geometry from Catia** window.

Figure 20: Import Geometry From Catia Window

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tetin file

Select the required path to store the Geometry file (*.tin) by clicking on the file browser icon.

In/Out Filters

Catia Input

Four CATIA input filters are provided here. By default the SHOW area is converted and the entities located in the NOSHOW area are ignored. Both PICK and NOPICK are usually enabled.

File Output

Suppresses specific entity classes, which are of no further importance for **ANSYS ICEM CFD**. Often CATIA models contain a lot of points and curves, which slow down the GUI and may even have to be deleted before meshing. Therefore, NO POINTS and NO CURVES are enabled by default while NO SURFACES is disabled by default.

Use CATIA Layer Filter

allows you to enter specific layer numbers to import. For instance specifying "17, 18" for **Filter args** would only import entities in these two layers and leave out other layers which may only contain construction geometry or the 2D drawing.

Other Options

The **Preview** option results in a roughly approximated conversion of the CATIA model with a large Triangulation Tolerance. Although approximation slows down the conversion, the resulting Tetin file is usually smaller and can be processed faster.

Sometimes trimming 2D boundary curves of CATIA faces exceeds the boundary of the underlying surface a little. These faces are suppressed by default. These suppressions are reported in the message window and if you want to force those faces to be contained in the Tetin file then you need to enable the **Ignore Range Errors** option.

To extract all boundary curves into the Tetin file you should enable **With Boundary Curves**.

Note

This option conflicts with the No Curves option so the latter should be disabled.

Note

This option may require a longer time to complete.

If **Loops instead of Faces** is enabled, planar faces are written as loops instead of Bspline surfaces in the Tetin file. Loops are easily meshed using the Shell Mesher.

Create Bounding Box will create a box around the converted model. The bounding box is automatically assigned a special part name.

Note

This option may require a longer time to complete.

If the faces of polyhedral solids in their original state are too complex to be represented by an unstructured domain, then by default they are represented by planer Bspline surfaces. If the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh polyhedral solids option is enabled, the surface mesher is invoked to convert complex shaped faces of polyhedral solids into an unstructured domain entity.

Estimate Maximum Size estimates the maximum size from the model dimension instead of using an infinite large value.

Take Layer from Shell/Skin/Solid only imports entities from layers containing these entity types. When enabled, it will not import entities from layers containing only curves or points.

Transform to active Axis System transforms geometry to the active local coordinate system.

Part Naming

You can select from the given four options:

Default

will directly transfer all the information of the CATIA model to the Tetin file. It will make the whole model into one part.

Part by Layer

will write a Tetin file by the layers defined in the CATIA. Entities kept in the same layers will be transferred into the same part.

Part by Color

will work the same way as layers. But it will translate the entities into part by color type instead of layer.

CATIAMIF-Parts

is chosen if the CATIAMIF interface was used on the CATIA model. It will give the name of the parts and mesh parameters set to the part defined by the CATIAMIF interface.

Entity Naming

This option is useful for giving names to the entities and parts.

Default

will write names as set in default.

CATIA-Names

will translate entities into Tetin files with the same names as defined in the CATIA files.

CATIAMIF-Names

is applicable if the CATIAMIF interface was used on the CATIA model. It will give the name of the parts as defined in the CATIAMIF interface.

Triangulation tolerance

sets up the global triangulation tolerance in ANSYS ICEM CFD (**Settings > Model**). This affects how the nurbs are rendered. It also affects mesh projection. Finer tri tolerance means slower rendering but more accurate projection. (Refer to Figure 76: Examples of Triangulation Tolerance (p. 87) for more information.)

DWG

The **DWG** option converts a DWG (AutoCAD Drawing) file to a Tetin file and loads it (Only 2D geometries).

Selecting the **DWG** option pulls up a file browser. Select the DWG file (*.dwg, *.dxf) to be imported and click **Open** to start translator.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

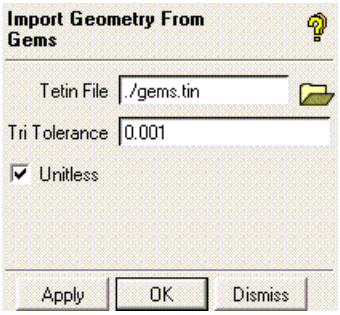
Another window will appear, where you have to specify path and name of the Tetin file which is converted from DWG file. After supplying the name and path, click **Open** to end the process.

GEMS

The **GEMS** option converts a GEMS file to a Tetin file and loads it.

Select the GEMS file to be imported and click **Open** to open the window shown below.

Figure 21: Import Geometry from GEMS window



Tetin File
select the required path to store the Tetin file by clicking on the icon.

Tri Tolerance
takes a default value of 0.001 if not otherwise specified.

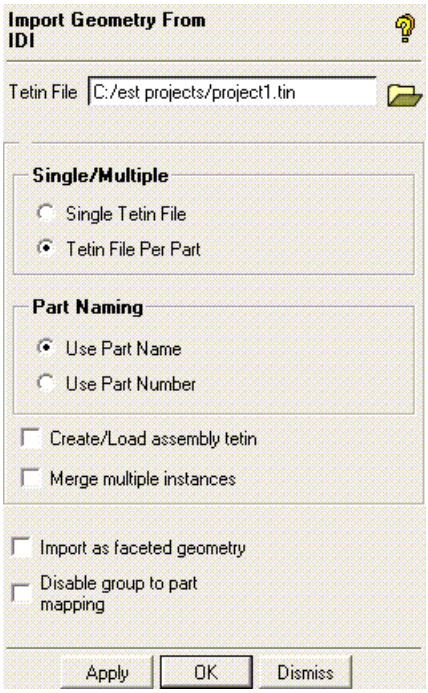
Unitless
if enabled, the exact unit of Gems geometry is not transferred to the Tetin file.

IDI

The **IDI** option allows you to import IDI files. Select the IDI geometry file to import. The following options are available for importing IDI geometry files.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22: Import Geometry From IDI window



Single Tetin File
creates one Tetin file for the IDI file. If this IDI file is an assembly file, then the Tetin file will contain all the parts.

Tetin File Per Part
creates one Tetin file for each part. If the IDI file is an assembly file, and **Create/Load assembly tetin** is enabled, then the Tetin file is also created for the entire assembly and is loaded into the GUI.

Part Naming
specifies whether part names or numbers are used.

Create/Load assembly tetin
if enabled, the assembly Tetin file will be created and loaded.

Merges multiple instances
if enabled, multiple instances of a part will be merged in one part file.

Import as faceted geometry
The IDEAS CAD data is actually high quality Bspline data. This is usually preferable, but this can add up to a large amount of data for large models. If this option is enabled, it will convert it to faceted data on import. This option is not recommended for use with patch based methods.

Disable group to part mapping
disables group to part mapping in the Tetin file.

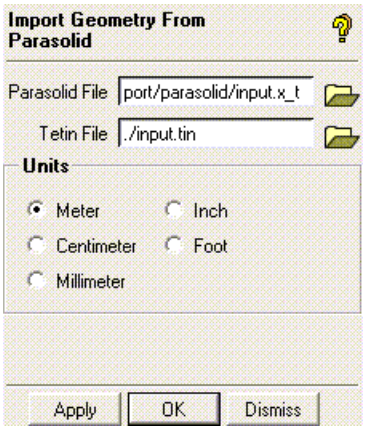
ParaSolid

The **ParaSolid** option converts a Parasolid model to a Geometry file (*.tin) and loads it.

The following options are available:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23: Import Geometry from Parasolid window

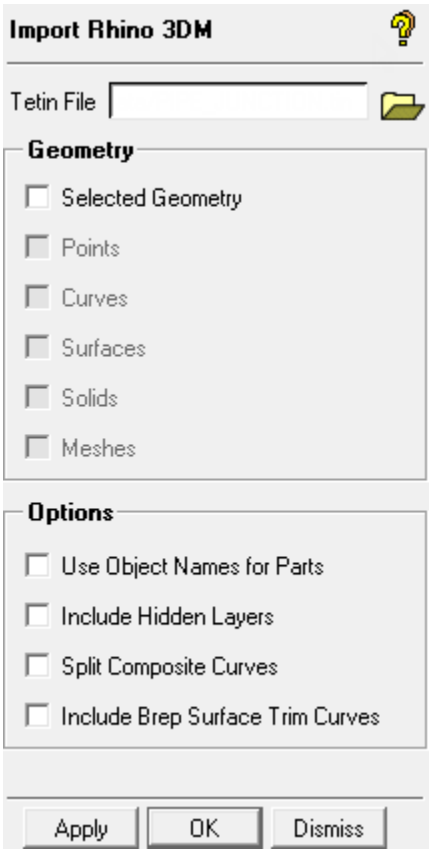


Parsolid File
specifies the Parasolid file.

Tetin File
specifies the output Tetin file.

Units
specifies the appropriate units of the geometry.

Rhino 3DM



The **Rhino 3DM** option converts a Rhino file (*.3dm) to a Geometry file (*.tin) and loads it. The following options for this conversion are available:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry

By default all the entities in the Rhino file will be imported. If **Selected Geometry** is clicked, then entities of different types can be specified for import:

- Points
- Curves
- Surfaces
- Solids

  This refers to manifold Boundary Representation (B-rep) solids.

- Meshes

  This refers to tessellated (STL-type) data.

Use Object Names for Parts

In Rhino, each object (point, curve, surface, etc.) may have a name associated with it. Enabling this option will use the Rhino object name as the Part Name. By default the name of the layer on which the object resides is used.

Include Hidden Layers

Enabling this option will import objects on hidden layers. By default, objects on hidden layers will not be imported.

Split Composite Curves

If enabled, composite curves will be split into their component curves and imported, instead of as a single curve.

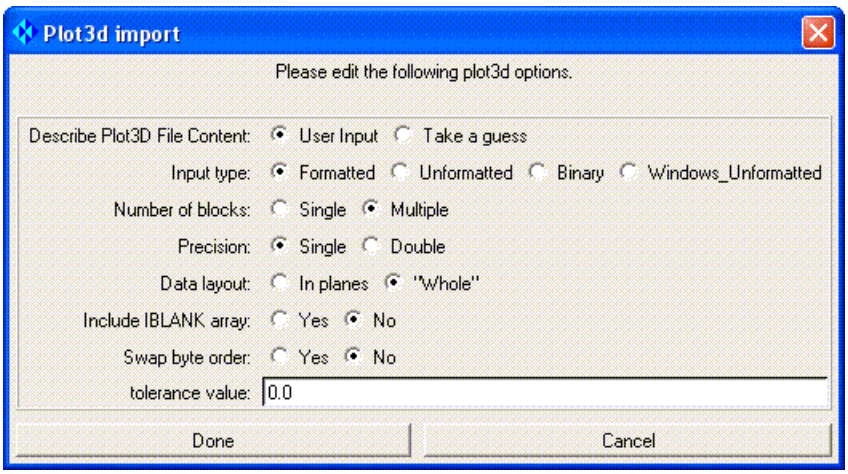
Include B-rep Surface Trim Curves

If enabled, the 3D spatial curves associated with the 2D parametric trim curves or a surface will be generated and imported, otherwise only the trimmed surface is imported.

Plot3d

The **Plot3d** option converts a Plot3d file to a Geometry file (*.tin) and loads it.

Figure 24: Import Geometry from Plot3d window


Plot3d import
Please edit the following plot3d options.
Describe Plot3D File Content: User Input Take a guess
Input type: Formatted Unformatted Binary Windows_Unformatted
Number of blocks: Single Multiple
Precision: Single Double
Data layout: In planes "Whole"
Include IBLANK array: Yes No
Swap byte order: Yes No
tolerance value: 0.0
Done
Cancel

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Describe Plot3D File Content
User Input allows you to specify file parameters.

Input type
Specify the input type. Windows_unformatted type refers to unformatted Plot3d meshes created on Windows. This option allows Unix platforms to read these files as well.

Number of blocks
Specify whether there is a single block or multiple blocks.

Precision
Specify either single or double precision.

Data layout
Specify whether data layout is in planes or not.

Include IBLANK array
Specify whether to include or omit IBLANK array.

Swap byte order
To toggle compatibility between platforms with little-endian or big-endian data storage.

STEP / IGES

The **STEP/IGES** option converts a STEP/IGES file to a Geometry file (*.tin) and loads it.

The following options are available for importing STEP/IGES geometry.

Figure 25: Import Geometry from Step or IGES window

Tetin File
specifies the required path to save the Tetin file by clicking on the icon.

Use Version 5.1 Step Translator
allows you to translate the file using ANSYS ICEM CFD 5.1 parameters.

Create part name from STEP/IGES file
If multiple STEP/IGES files are opened at once, this option will create the part names in the converted geometry file from the STEP/IGES files.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge geometry files after conversion
If multiple STEP/IGES files are opened at once, this option will merge the multiple geometry files after conversion.

Ignore units
if enabled, the units specified in the import file will be disregarded and meters are assumed to be the length unit.

Use healing
if enabled, the Parasolid Kernel attempts healing on the IGES geometry (bodies, surfaces, and loops) before sending it to the IGES translator.

Formatted Point Data

The **Formatted point data** option converts formatted point data into the ANSYS ICEM CFD format. The data is read in as Bspline data.

In this format, each surface's data is represented in the following way: The first line contains two numbers, the number points for each curve, and the number of curves for each surfaces. The remaining lines represent the points of the surface. An example is shown below:

2 4

0.0682618 9.731237 0.5112982

-0.2821764 19.30035 0.5278823

-1.046131 9.840971 1.598944

-1.646277 19.23213 1.157967

-1.866522 9.97312 2.660025

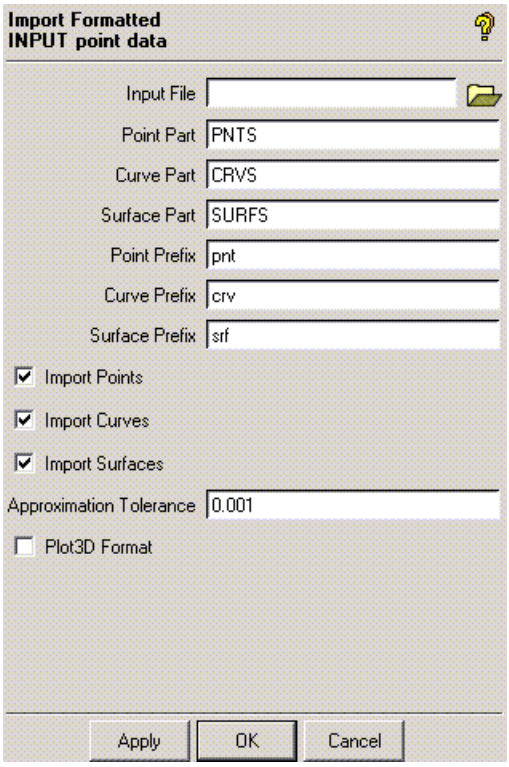
-2.986559 19.06998 1.786853

-2.552526 10.16115 3.688797

-4.276251 18.82248 2.374612

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26: Import Geometry from Formatted Point Data window



Reference Geometry

The **Reference Geometry** option allows you to use an existing Tetin file as a reference to map the mesh setup and other parameters to the currently loaded geometry file. This is important in parametric modeling, as the setup that is done for one Tetin file can be reused for subsequent design changes.

The parameters that will be read from the reference geometry file include:

- Global Meshing parameters
- Entity meshing parameters
- Prism meshing parameters
- Part names of entities
- Dormant curves and points
- Density regions
- Connector definitions
- Geometry subsets
- Periodicity

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Mesh

Figure 27: Import Mesh options

The **Import Mesh** menu allows you to import mesh from the following formats:

From ANSYS
From Abaqus
From CFX
From CGNS
From Fluent
From LS-DYNA
From Nastran
From Patran
From Plot3d
From Starcd
From STL
From TecPlot
From Vectis

From ANSYS

The **From ANSYS** option allows you to import an ANSYS mesh file into ANSYS ICEM CFD. Select a *.cdb file to import.

From Abaqus

The **From Abaqus** option allows you to import an Abaqus mesh file into ANSYS ICEM CFD.

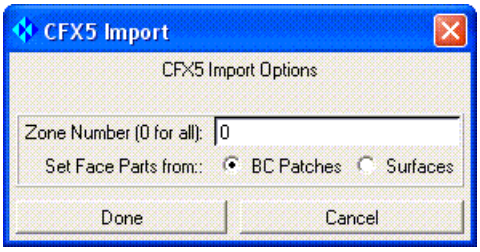
From CFX

The **From CFX** option allows you to import a CFX5 mesh file into ANSYS ICEM CFD.

After selecting the CFX5 *.def file, you need to choose the translation option shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28: Import CFX Options

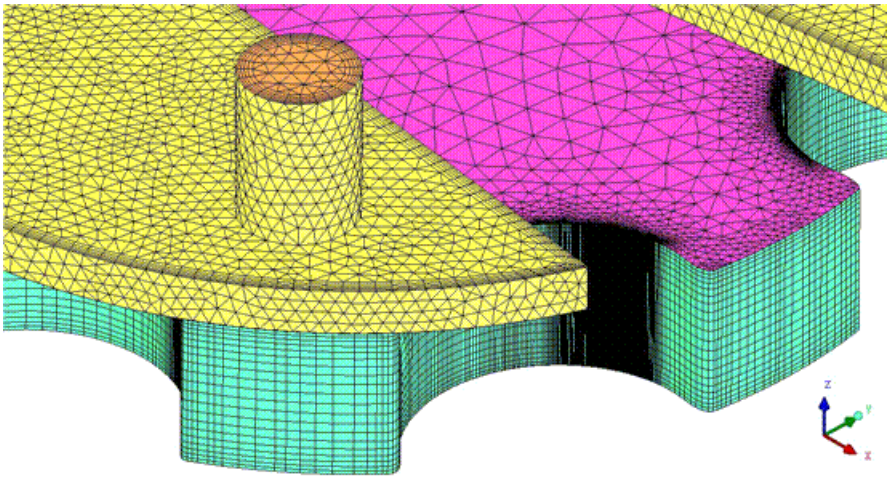


Set Face Parts from

BC Patches

creates a Part for each different boundary condition (inlet, outlet, wall, Domain-Interface-1-side-1, etc.) as shown in Figure 29: Set Face Parts from BC Patches Option (p. 38).

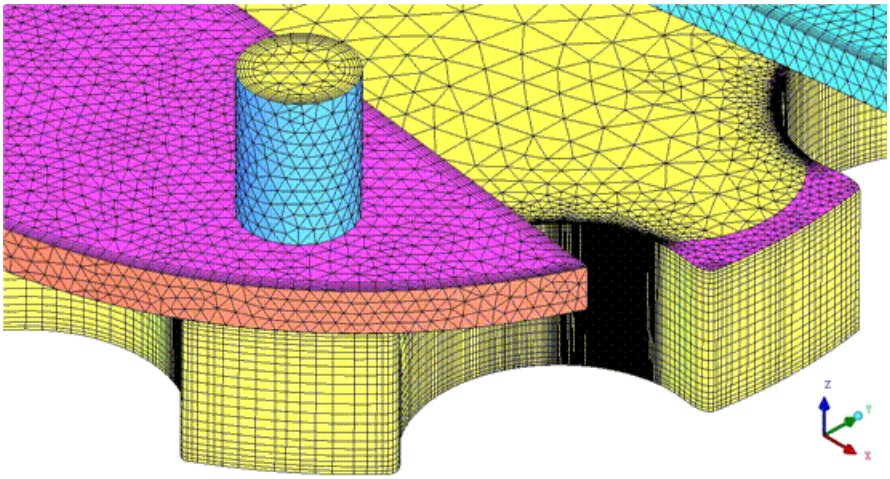
Figure 29: Set Face Parts from BC Patches Option



Surfaces

creates a part from each surface (surface 1, surface 2, etc.) as shown in Figure 30: Set Face Parts from Surfaces Option (p. 38).

Figure 30: Set Face Parts from Surfaces Option



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click **Done** to complete the process.

Note

CFX5 mesh import disallows duplicate nodes and faces. Internal walls should not be split, because this introduces duplicate nodes and faces and the connectivity between the faces may be incorrect. Use Reorient Mesh > Reorient Consistent on all thin surfaces prior to export to avoid inconsistencies in the orientation of face elements.

From CGNS

The **From CGNS** option allows you to import a CGNS mesh file into ANSYS ICEM CFD. You need to specify whether the input mesh file is structured or unstructured mesh.

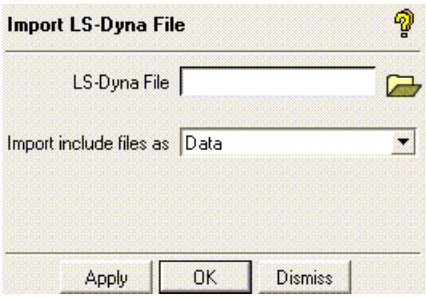
From Fluent

The **From Fluent** option allows you to import a Fluent mesh file into ANSYS ICEM CFD. Select a *.cas file to import.

From LS-DYNA

The **From LS-DYNA** option allows you to import an LS-DYNA mesh file into ANSYS ICEM CFD.

Figure 31: Import LS-DYNA Options



Import include files as

INCLUDE files can be read in as links or as actual data.

Note

If the INCLUDE files are read in as data, they will be written to the assembly file and the original file structure will not be preserved.

Note

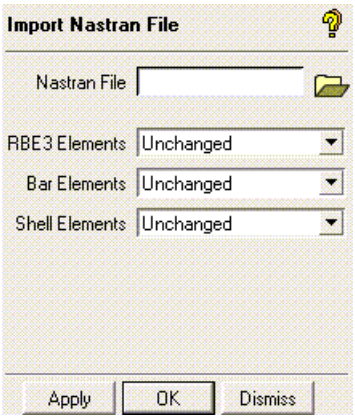
Any unsupported LS-DYNA cards will be written to a temp file named “unsupported.k” in the working directory, and will be appended to the input file of this project if it is written out.

From Nastran

The **From Nastran** option allows you to import a Nastran mesh file into ANSYS ICEM CFD.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

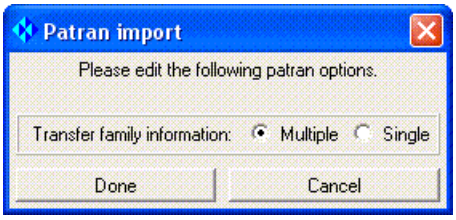
Figure 32: Import Mesh from Nastran Options



From Patran

The **From Patran** option allows you to import a Patran mesh file into ANSYS ICEM CFD.

Figure 33: Import Mesh from Patran Options



Transfer family information

Multiple
converts each Patran group (PID) into an ICEM CFD part.

Single
brings the Patran groups all in as one part.

From Plot3d

The **From Plot3d** option allows you to import a Plot3d mesh file into ANSYS ICEM CFD.

Plot3d files do not provide a topology, so ANSYS ICEM CFD will recompute a topology. Mesh will be converted to an unstructured mesh by default. The subfaces, edges, and vertices belonging to the re-computed topology will added to separate parts.

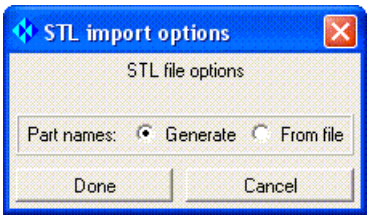
From Starcd

The **From Starcd** option allows you to import a STAR-CD mesh file into ANSYS ICEM CFD. Select any one of the *.cel, *.vrt, or *.inp files with the same file name.

From STL

The **From STL** option allows you to import an STL mesh file into ANSYS ICEM CFD. Note that the parts defined in the STL file will also be imported. Select the format for the part names and click **Done** to finish the operation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Part names

Generate
generates part names of the format **`Filename.PartName`**, where **`PartName`** is the part name defined in the STL file.

From file
retains the part names as defined in the STL file.

> **Note**
>
> This option is valid only for STL files in ASCII format. For STL files in binary format, the part names of the format **`Filename.PartName`** will be generated as described.

From TecPlot

The **From TecPlot** import filter allows you to read CFX files and convert the mesh data to ANSYS ICEM CFD unstructured mesh.

From Vectis

The **From Vectis** option allows you to import a Vectis mesh file (*.sdf) into ANSYS ICEM CFD.

> **Note**
>
> This option is available only on Linux systems.

Export Geometry

Figure 34: Export Geometry Options

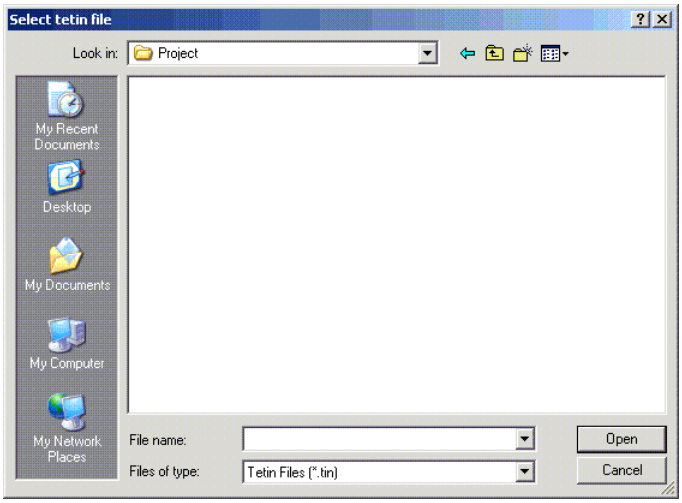
The **Export Geometry** option allows you to export geometry in the following formats.
To IGES
To Parasolid
To Rhino 3DM
To STL

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To IGES

The **To IGES** option allows you to export geometry in IGES format. After selecting **To IGES**, select the geometry file (*.tin) that you want to import in the **Select tetin file** window.

Figure 35: Export to IGES File



After selecting the geometry file, supply the name and the path of for the IGES file and click **Save** in the **Select iges file** window to complete the export process.

Note

Faceted data cannot be exported to an IGES file. If faceted data is found in the geometry file, it will not be translated.

To Parasolid

The **To Parasolid** option converts geometry in ANSYS ICEM CFD to a Parasolid file.

Select **To Parasolid** and then select the tetin file to export in the **Select tetin file** window. After selecting the geometry file, supply the name and the path of for the IGES file and click **Save** in the **Select ps file** window to complete the export process.

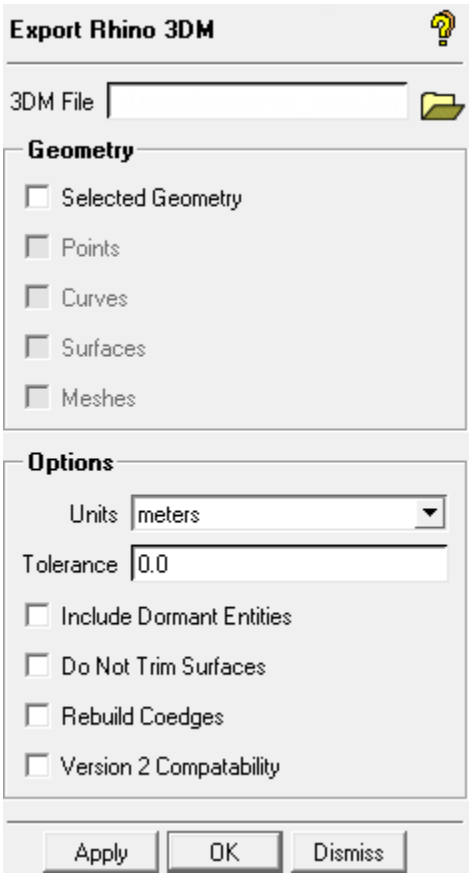
Files with an asterisk (*) indicates these files are OK to select.

Note

Faceted data cannot be exported to a Parasolid file. If faceted data is found in the geometry file, it will not be translated.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Rhino 3DM



The **To Rhino 3DM** option converts geometry in **ANSYS ICEM CFD** to a Rhino 3DM file format (*.3dm). Select the geometry file (*.tin) to be exported. The following options are available.

Geometry

By default all the entities in the geometry file will be exported. If **Selected Geometry** is clicked, then entities of different types can be specified for export:

- Points
- Curves
- Surfaces
- Meshes

Units

Specify the units of measurement of the file.

Tolerance

This is the absolute tolerance value passed to the Rhino file. It is used in Rhino for operations such as trimming and intersecting.

Include Dormant Entities

By default, any dormant entities (typically points and curves) created will not be exported to Rhino. If this option is enabled, the dormant entities will also be exported.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

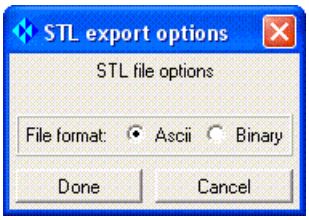
Do Not Trim Surfaces
If this option is enabled, any trim curves are removed, so that trimmed surfaces will be exported to Rhino as untrimmed.

Rebuild Coedges
If enabled, the coedges (3D spatial curves) will be rebuilt from the 2D parametric curves (pcurves) by projection prior to exporting. This sometimes improves the quality of the exported 3DM file, particularly when the Tetin file has relatively large tolerances.

Version 2 Compatibility
The default will export the data to a Version 3 3DM (Rhino) file, which cannot be read by Rhino Version 2.xx. If this option is enabled, then the 3DM file will be version 2 compatible.

To STL

The **To STL** option converts geometry in ANSYS ICEM CFD to the STL format. Select the appropriate file format and click **Done** to finish the operation.



File format
specifies the format for the STL file.

Ascii
exports the STL file in ASCII format.

Binary
exports the STL file in binary format.

Export Mesh

Figure 36: Export Mesh Options

Mesh generated in ANSYS ICEM CFD can be exported to the following formats:
To Abaqus
To ANSYS

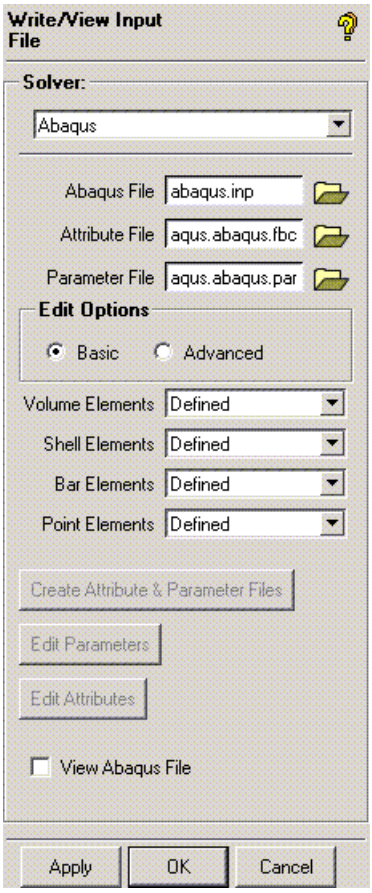
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Autodyn
To Exodus
To Ideas
To LS-DYNA
To Nastran
To Patran
To Radioss
To FE Modeler
Write STL file
Spectral Elements

To Abaqus

The **To Abaqus** option allows you to export the mesh in Abaqus format. For information about the Abaqus interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD CFD Output Interfaces information, and click ABAQUS in the Table of Supported Solvers.

Figure 37: Abaqus Export Options

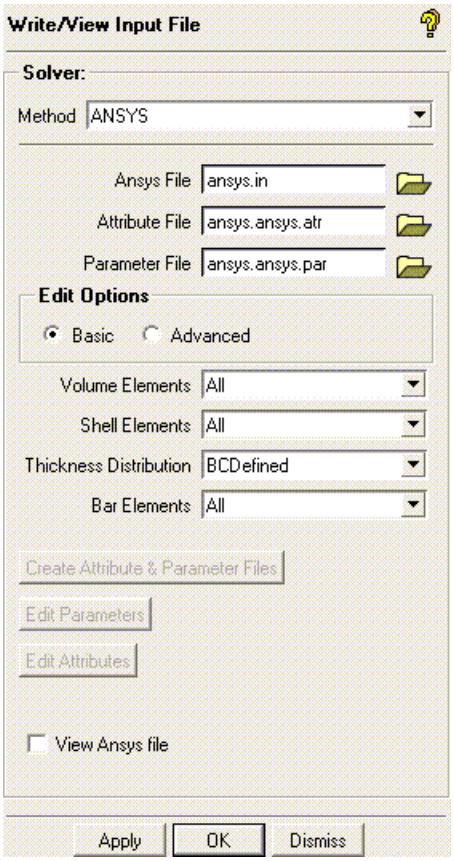


To ANSYS

The **To ANSYS** option allows you to export the mesh in ANSYS format. For information about the ANSYS interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click ANSYS in the Table of Supported Solvers.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38: ANSYS Export Options



To Autodyn

The **To Autodyn** option allows you to export the mesh in Autodyn format. Information for this option can be found in the Solve Options > Autodyn Options > Write/View Input File section.

Note

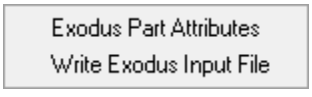
This solver format supports multiblock structured or unstructured mesh. If a multiblock domain is loaded, the multiblock.info interpreter will be used. For unstructured mesh, the LS-DYNA interpreter will be used to create the unstructured mesh.

To Exodus

For information about the Exodus interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click EXODUS in the Table of Supported Solvers.

The options available are:

Figure 39: Exodus Options

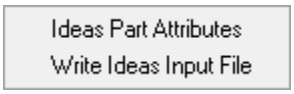


ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To Ideas

The **To Ideas** option allows you to export the mesh in Ideas format. For information about the Ideas interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click IDEAS in the Table of Supported Solvers.

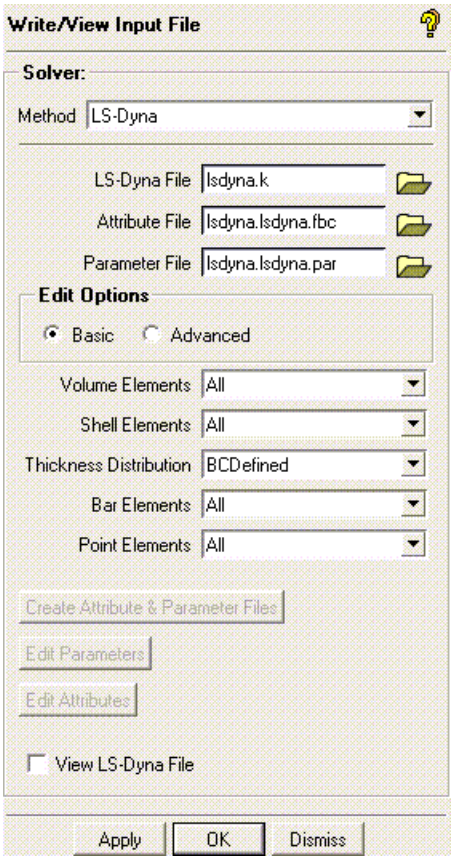
Figure 40: Ideas Options



To LS-DYNA

The **To LS-DYNA** option allows you to export the mesh in LS-DYNA format. For information about the LS-DYNA interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click LS-DYNA in the Table of Supported Solvers.

Figure 41: LS–Dyna Export Options

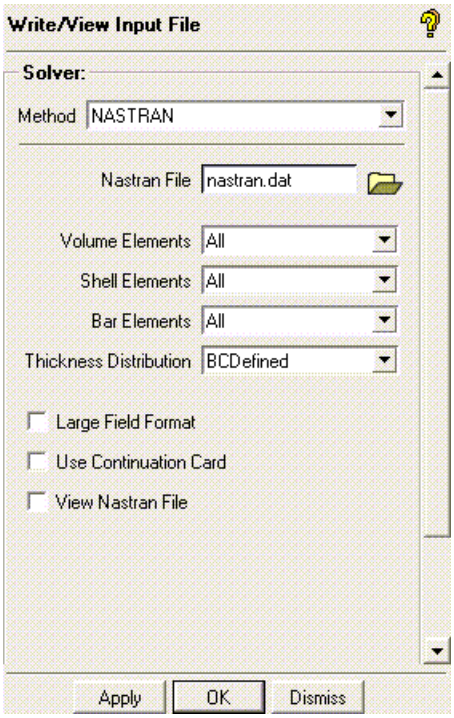


To Nastran

The **To Nastran** option allows you to export the mesh in Nastran format. For information about the Nastran interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click NASTRAN in the Table of Supported Solvers.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 42: Nastran Export Options



To Patran

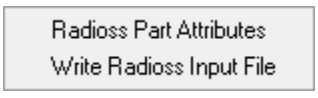
The **To Patran** option allows you to export the mesh in Patran format. For information about the Patran interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click PATRAN in the Table of Supported Solvers.

Figure 43: Patran options window

To Radioss

The **To Radioss** option allows you to export the mesh in Radioss format. For information about the Radioss interface, select the **Output Interfaces** option in the **Help** menu to open the ANSYS ICEM CFD Output Interfaces information, and click RADIOSS in the Table of Supported Solvers.

Figure 44: Radioss options window



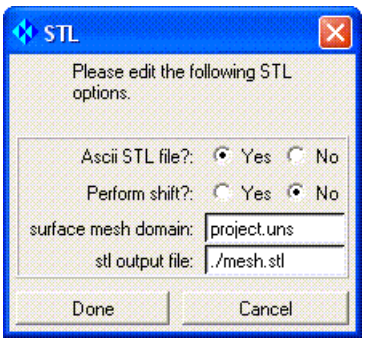
To FE Modeler

The **To FE Modeler** option allows you to export the mesh to a *.dat file (similar to the Nastran format) which can be imported into FE Modeler. For v15.0, you can also save the *.uns file and read it in FE Modeler.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write STL file

The **Write STL File** option allows you to write mesh data as faceted data in STL format which can be read by any other solver/software. Specify the parameters as appropriate and click **Done** to finish the operation.



Ascii STL file?
allows you to write the STL file in ASCII or binary format.

Perform shift?
checks if a vertex extends into negative space. When enabled, the whole mesh will be shifted by the amount the vertex extends into negative space.

Surface Mesh domain
specifies the surface mesh domain file.

STL output file
specifies the output STL file.

Spectral Elements

The **Spectral Elements** option allows you to create spectral elements. Select the elements to be created into spectral elements. Select the **Order** and the **Bunching law** to be used. The following Bunching laws are available:

- Gauss-Lobatto-Legendre
- Chebyshev-Lobatto
- Legendre

Replay Scripts

ANSYS ICEM CFD makes it possible for users to write their own specialized scripts to customize **ANSYS ICEM CFD** or run complex operations in batch mode. A script is basically a program written in the Tcl/Tk language. There are a number of standard references for this language, and more information that can be found at http://www.scriptics.com/.

The **Replay Scripts** menu item opens a submenu of options as shown in Figure 45: Replay Options (p. 50).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45: Replay Options

Replay Control

This option helps you create script files by performing operations in **ANSYS ICEM CFD** and recording the equivalent Tcl/Tk commands in a Replay file. The user can modify or run this Replay file as a script file. This gives opportunity for users unacquainted with Tcl/Tk to also use scripting.

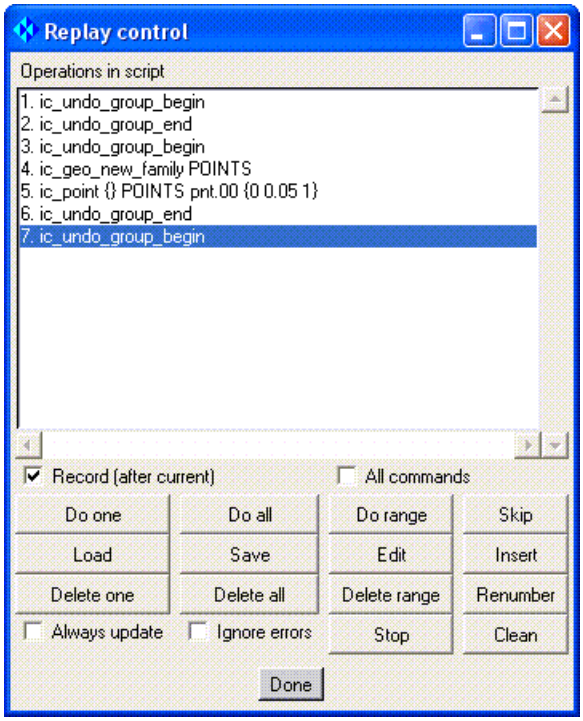
Note

If you are using **ANSYS ICEM CFD** in Workbench and want to step through Workbench Input Parameters you have created, you must open the Workbench Replay Control from the One-click menu. For more information, See Workbench Integration in the *ANSYS ICEM CFD User's Manual*.

The example shown in Figure 46: Replay Control window (p. 50) contains the script commands recorded while the following operations were performed in **ANSYS ICEM CFD** :

Geometry > Create Point > Screen Select.
New part name is POINTS.
Screen select a location on the loaded Geometry file.

Figure 46: Replay Control window



The **Replay control** window can be minimized at any time.

Record (after current)

To record commands in the Replay file, toggle this option ON. Commands will be appended.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

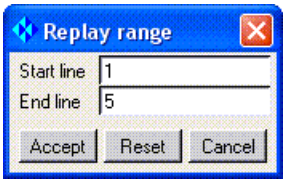
All commands
To record all the commands that you or ANSYS ICEM CFD performs, toggle this option ON.

Do one
While running the script, clicking this button will perform only one step, or one line of the Replay file.

Do all
ANSYS ICEM CFD will run the whole Replay file when this option is selected.

Do Range
To run a specific subset of the commands in the Replay file, select this option. The **Replay Range** window opens where you select the *Start line* and *End line* numbers, as shown in Figure 47: Replay Range window (p. 51).

Figure 47: Replay Range window



Skip
To ignore the command line where the cursor lies, select this option. The cursor will move to the next line.

Load
To open a previously stored Replay file (`*.rpl`) in the **Replay Control** window, select this option. The **File Selection** window opens where you choose the Replay file to be loaded.

Save
Saves the Replay file in a location specified in the **File selection** window.

Edit
Opens the Replay script in your text editor (e.g. Notepad) where you to modify the script. You will be prompted to save your work when you close the editor.

Insert
Inserts one line in the script so that you can enter comments or commands.

Delete one
Removes the selected line of command from the Replay file.

Delete all
Clears all the commands from the **Replay Control** window.

Delete range
Deletes a specific range of commands from the Replay file.

Renumber
If you have added or deleted commands manually, this button re-sequences the command line numbers.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Always update
Updates the model after each step. When this is disabled, the replay script will be run in batch mode and the model will be updated at the end. You can enable this option to see the step by step updates, however, you can disable it to more rapidly generate the model.

Ignore errors
When disabled (default), the replay control will stop executing the replay script when an erroneous command is reached. When enabled, the replay control will skip the erroneous command and continue to execute the subsequent commands in the replay script.

Stop
Halts an ongoing Replay of your script file.

Clean
Filters through the Replay file and removes unnecessary lines. You can adjust this clean filter by right clicking on the Replay window and choosing **Edit Filter**. Simply enter the ic_commands that you want removed. There is also the right click option to **Undo Clean**, which will undo the actions of the clean filter.

Note

If an Undo action was performed while recording the Replay file, then the user needs to manually remove the appropriate lines from the script (starting with `undo_begin` and ending with the `undo_end` command). The Clean filter does not take care of this.

Done
Closes the **Replay Control** window. You will be prompted to save the Replay file before closing the window if it is not saved.

Load script file
The **Load script file** option opens a dialog box where you select a replay script file (`*.rpl`) to display in the **Replay Control** window.

Run from script file
The **Run from script file** option opens a dialog box where you select a previously written script and initiate automatic replay.

Note

In order to know what commands are available to you to do useful things like load meshes and run applications, a set of functions that begin with the letters **ic_** that can be called from scripts are documented in the Programmer's Guide. The user is discouraged from reading through the tcl files in *$ICEM_ACN/bin/med* and using undocumented functions because they might change at any time.

Recording scripts
The **Recording scripts** option starts/stops the recording of commands.

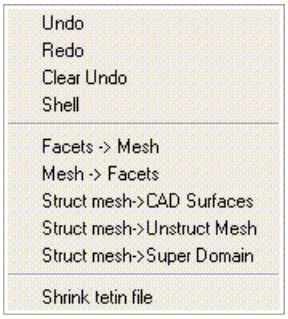
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exit

The **Exit** option closes the **ANSYS ICEM CFD** application. You will be prompted to save any new work in your project.

Edit Menu

Figure 48: Edit menu



The **Edit** menu has the following options:

Undo
Redo
Clear Undo
Shell
Facets > Mesh
Mesh > Facets
Struct Mesh > CAD Surfaces
Struct Mesh > Unstruct Mesh
Struct Mesh > Super Domain
Shrink Tetin File

Undo

The **Undo** option undoes the last operation. For a few operations like smoothing, refinement, coarsening, conversion, etc., where the whole mesh is changing, only **one Undo** operation is possible.

Note

Operations related to display cannot typically be undone.

Note

When using **Undo** for blocking operations, the index control may be affected because the active state of the index control may be modified by the Undo operation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Redo

The **Redo** option redoes the previous **Undo** operation. You can only redo operations that have been undone.

Note

Once a new command is performed; all operations undone are lost from the undo/redo history.

Clear Undo

The **Clear Undo** option clears up the memory used by the **Undo** history. This option is most useful after performing an operation (such as "Add to Part" , "Change Type", or "Split/Redistribute Prisms") on a large number of elements because the **Undo** needs to store the state both before and after the operation in memory.

Note

The **Clear Undo** option frees up the memory for further operations, but you will not be able to undo the particular operation any more.

Shell

The **Shell** option opens a new terminal in the current working directory.

Facets > Mesh

The **Facets > Mesh** option will convert the faceted geometry data into unstructured mesh format.

Mesh > Facets

The **Mesh > Facets** option will convert unstructured mesh format into faceted geometry data.

Struct Mesh > CAD Surfaces

The **Struct Mesh > CAD Surfaces** option will convert structured mesh format into CAD surfaces.

Struct Mesh > Unstruct Mesh

The **Struct Mesh > Unstruct Mesh** option will convert structured mesh format into unstructured mesh format.

Struct Mesh > Super Domain

The **Struct Mesh > Super Domain** option will convert structured mesh format into a superdomain file. In the popup window that appears, highlight the domains with the specified project prefix that are to be converted to a superdomain.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shrink Tetin File

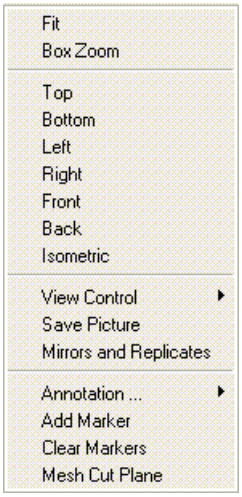
The **Shrink tetin file** option removes header lines, comment lines, and unnecessary definitions in the Tetin files, which can reduce the size of the Tetin file significantly. The comment lines include the date and software version of the last modification.

Selecting this option will automatically open a browser window. Select the desired Tetin file. The number of lines removed will appear in the Message window. From the browser window, you can either save the file with a new name or overwrite the original file.

View Menu

The **View** menu contains options for choosing different orientations of the model in the graphics display.

Figure 49: View Options



The different options are shown in Figure 49: View Options (p. 55).

- Fit
- Box Zoom
- Top
- Bottom
- Left
- Right
- Front
- Back
- Isometric
- View Control
- Save Picture
- Mirror and Replicates
- Annotation
- Add Marker
- Clear Markers
- Mesh Cut Plane

Fit

The **Fit** option scales the model so that it fits in the graphics window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box Zoom

The **Box Zoom** option will prompt you to click and drag a rectangular region which is then zoomed in to fill the graphics window.

Top

The **Top** option orients the model so that you see it parallel to the XZ-plane, looking from the positive Y-axis.

Bottom

The **Bottom** option orients the model so that you see it parallel to the XZ-plane, looking from the negative Y-axis.

Left

The **Left** option orients the model so that you see it parallel to the YZ-plane, looking from the negative X-axis.

Right

The **Right** option orients the model so that you see it parallel to the yZ-plane, looking from the positive X-axis.

Front

The **Front** option orients the model so that you see it parallel to the XY-plane, looking from the positive Z-axis.

Back

The **Back** option orients the model so that you see it parallel to the XY-plane, looking from the negative Z-axis.

Isometric

The **Isometric** option orients the model so that you see a 3D pictorial representation of the model is shown.

Note

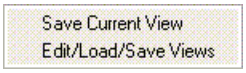
One of the six orthographic views or the isometric view may be easily selected by clicking on one of the points of the Display Triad in the graphics window.

View Control

The **View Control** options allow you to save or edit the current view.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 50: View Control Options



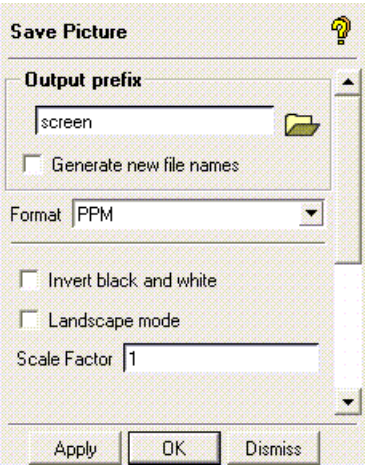
Save Current View
allows you to save the current view.

Edit/Load/Save Views
allows you to edit and save different views. You can save the views to an external file for use with any project. The views can also be saved with the current project file when the **Automatically Save Views to Project** option in the **Settings - General** DEZ is enabled (see the General (p. 70) section). The **View Control** appears in the bottom-right corner of the screen.

Save Picture

The **Save Picture** option allows you to save different views of the model in different picture formats. The format can be selected from the **Save Picture** DEZ shown in Figure 51: Save Picture DEZ (p. 57).

Figure 51: Save Picture DEZ



Output Prefix
specifies the name of the picture file you want to save in the current directory (where project file exists). Alternatively, you can browse to any other directory by clicking on the icon.

Generate new file names
toggles the generation of output files with new names each time the function is used.

Format
specifies the picture format. You can select any of the following picture formats: PostScript, PS from screen, PPM, X11, TIFF, GIF, JPEG, and VRML.

The following options are available for the PS from screen, PPM, X11, TIFF, GIF, and VRML formats:

Invert black and white
controls the foreground/background color. When enabled, the background color in the saved image will be changed from black to white.

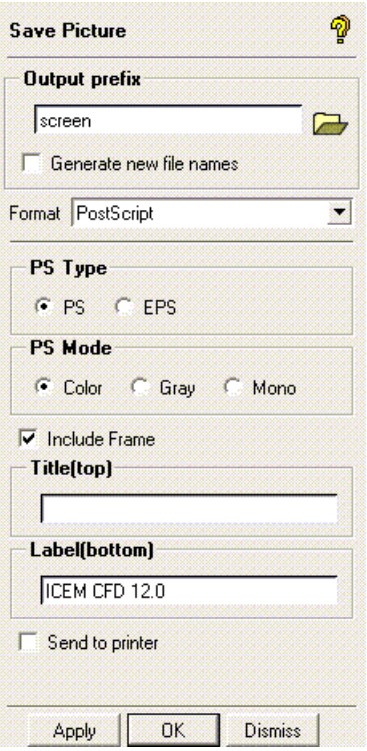
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Landscape mode
controls the orientation of the saved image. When enabled, the image will be saved in landscape mode; else the image will be saved in portrait mode.

Scale Factor
scales the image taken with respect to the actual size appearing in the graphics window. Example: A Scale Factor of 2 doubles the size of the picture taken.

Quality (for JPEG format only)
specifies the quality level, which will determine the file size.

Figure 52: PostScript Format Options



The following options are available for the PostScript format:

PS Type
specifies the PS type. The **PS** format is suitable for operating systems other than Windows, like UNIX. The **EPS** format is suitable for the Windows operating system.

PS Mode
specifies the color mode for the image saved. **Color** maintains the actual colors of the picture. **Gray** converts all the colors to gray scale. **Mono** converts the picture to black and white.

Include Frame
shows the border of the picture in a different color than the background.

Title (top)
specifies the title for the image. This can be viewed at the top of the postscript file.

Label (bottom)
specifies the label which will appear at the bottom of the postscript file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

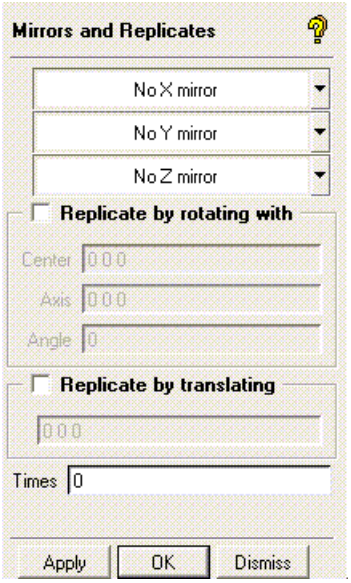
Send to printer
allows you to print the picture after saving.

Mirror and Replicates

The **Mirrors and Replicates** DEZ contains options that allow you to create a mirror image of a model.

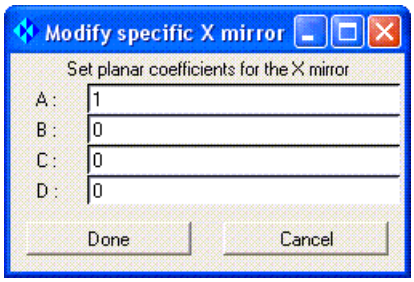
> **Note**
>
> Mirrors and replicates are not real objects. They are only for display purposes and therefore cannot be selected or have operations performed on them.

Figure 53: Mirror and Replicates DEZ



X, Y, Z Mirror
contains options for defining the mirror plane. You can use the standard defined mirror planes (**normal** or **outward** in the X, Y, or Z directions) or define a specific plane in the required direction using the **Modify specific mirror** dialog (seeFigure 54:**Modify specific mirror** Dialog (p. 59)).

Figure 54: Modify specific mirror Dialog



Replicate by rotating with
enables the creation of replicates by rotation. Specify the **Center**, **Axis**, and **Angle** of rotation to create the replicates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

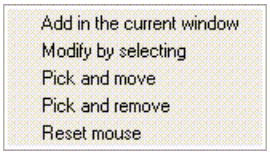
Replicate by translating
enables the creation of replicates by translation. Specify the distances in the X, Y, and Z directions for the translation.

Times
indicates the number of replicates.

Annotation

Annotations are graphical objects (such as thumbnail pictures or some explanatory text) that are defined independently of the data being displayed, and are generally fixed in position in the window.

Figure 55: Annotations Options

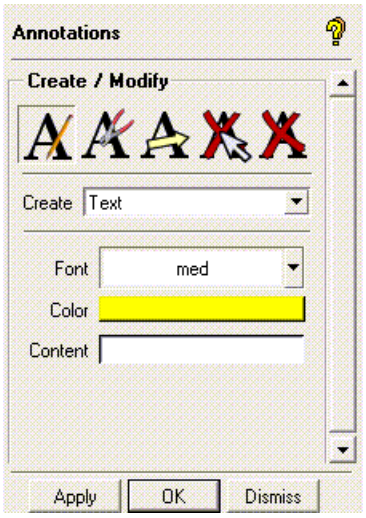


The available annotation options are as follows:
Add in the current window
Modify by selecting
Pick and move
Pick and remove
Reset mouse

Add in the current window

The **Add in the current window** option allows you to add annotations to the currently displayed data.

Figure 56: Create/Modify Annotation window



Create
allows you to create the following types of annotations:

- Text
- Utf Text

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Lines
- Box
- Circle
- Polygon
- Marks
- Image
- Colormap Bar

The options available are:

Text font
specifies the font size to use for text annotations.

Line width
specifies the line width, in pixels, used for drawing line, circle, box, and polygon annotations.

Arrow type
determines whether an arrow will be drawn at the start and/or end points of the line.

Symbol type
specifies the type of symbols used in mark annotation.

Symbol size
specifies the size of marker annotations.

Color
specifies the color of the annotation.

Fill
enables the filling of box, circle, polygon, and mark annotations.

Modify by selecting

The **Modify by selecting** option allows you to modify features of specific annotations, such as size, text, color, and font.

Pick and move

The **Pick and move** option allows you to pick and move specific annotations.

Pick and remove

The **Pick and remove** option allows you to pick and remove specific annotations.

Reset mouse

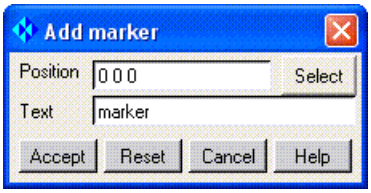
The **Reset mouse** option allows you to reset the mouse.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add Marker

The **Add Marker** option allows you to display text in the GUI using the **Add marker** dialog (see Figure 57: Add Marker Dialog (p. 62)).

Figure 57: Add Marker Dialog



Position
specifies the coordinates for the position of the marker.

Select
allows you to select the position of the marker from the existing geometric information.

Text
specifies the text to be displayed as a marker.

Accept
creates given text at a given location as a marker.

Reset
resets the position and text to the previously created marker.

Cancel
cancels the command and closes the **Add Marker** dialog.

Help
launches the Help manual.

Clear Markers

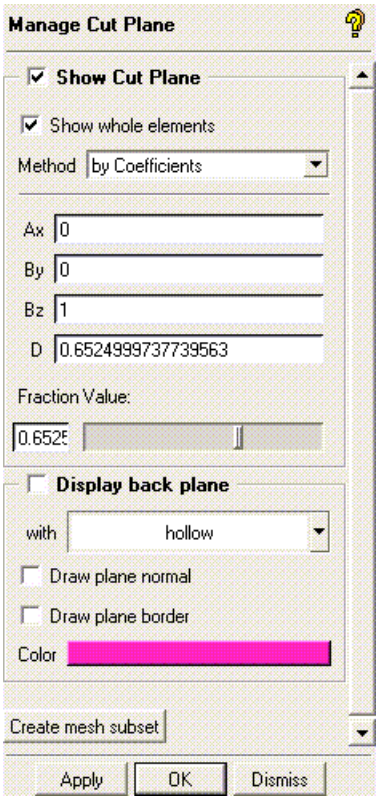
The **Clear Marker** option allows you to clear all the created markers from the GUI.

Mesh Cut Plane

A cut plane is used to visualize results on a plane cut through the three dimensional model. Results are viewed on the cut plane as well as on the 2D Dynamic window. The cut plane may be defined in several ways depending on the application. These options are described here in detail.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 58: Manage Cut Planes DEZ



Show Cut Plane
enables the display of the cut plane.

Show whole elements
enables the display of the elements at the borders of the cut plane.

Method
contains options for defining the cut plane.

By Coefficients
defines the cut plane by the equation of the normal vector. The equation is of the form: Ax + By + Cz + D = 0. Specify the coefficients A, B, C, and D in the fields **Ax** , **By**, **Bz**, and **D**.

By Point and Normal
defines the cut plane by a point and the normal. Specify the Global Cartesian coordinates in the X, Y, and Z direction for the point (**Pn**) and the X, Y, and Z components of a unit vector normal to the desired cut plane (**NX**, **NY**, and **NZ**).

By Corner Points
creates a cut plane passing through the three specified points. You can enter the Cartesian coordinates for **Pt1**, **Pt2**, and **Pt3**.

By 3 Points
creates a cut plane passing through the three locations selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Move or Rotate
allows you to interactively move and/or rotate a previously defined cut plane. Use the left mouse button to rotate the cut plane about the normal axis and the middle mouse button to move the cut plane. Click the right mouse button to end the interactive movement of the cut plane.

Middle X plane
positions the cut plane at the middle of the geometry in the X-direction.

Middle Y plane
positions the cut plane at the middle of the geometry in the Y-direction.

Middle Z plane
positions the cut plane at the middle of the geometry in the Z-direction.

Fraction Value
specifies the location of the plane.

Display back plane
enables the display of the back plane.

Draw plane normal
enables the display of the normal direction of the plane.

Draw plane border
enables the display of the border of the back plane.

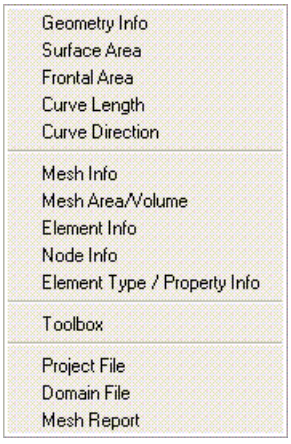
Color
specifies the color of the back plane. To select a different color, click on the color bar and select a color from the menu that appears.

Create mesh subset
creates a mesh subset of cut plane elements.

Info Menu

The **Info** menu provides useful information related to the model like curve length, node information, distance between two points, etc.

Figure 59: Info Options



The options available are as follows:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Info
Surface Area
Frontal Area
Curve Length
Curve Direction
Mesh Info
Mesh Area/Volume
Element Info
Node Info
Element Type/ Property Info
Toolbox
Project File
Domain File
Mesh Report

Geometry Info

The **Geometry Info** option prints the number of surfaces, curves, material points and parts existing in the model in the message window.

Surface Area

The **Surface Area** option prints information (like surface names, total area, etc.) for the selected surfaces in the message window.

Frontal Area

The **Frontal Area** option prints information about the area of the model in front of the screen in the message window.

Curve Length

The **Curve Length** option prints information (like curve name, curve length, etc.) for the selected curves in the message window.

Curve Direction

The **Curve Direction** option will prompt you to select curves. The curve direction will be displayed for each selected curve. The curve direction is the direction of increasing curve parameter, from 0 to 1.

Note

The curve direction can be reversed using the **Reverse Direction** option in the **Modify Curve(s)** DEZ. Curve direction may affect some meshing and geometry functions that utilize curve end locations.

Mesh Info

The **Mesh Info** option prints the following information about the mesh in the message window:

- Total number of elements

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Total number of nodes
- The minimum (Min) and maximum (Max) coordinates of the bounding box.
- Number of elements by type
- Number of elements by parts
- Maximum edge sides

  This is the maximum edge distance between nodes of the mesh. The nodes will be placed in the subset that is listed.
- Minimum edge sides

  This is the minimum edge distance between nodes of the mesh. The nodes will be placed in the subset that is listed.

Mesh Area/Volume

The **Mesh Area/Volume** option prints the area and volume of the mesh elements by type, and the total area and volume in the message window. For example:

```
Area of 5534 TRI_3 is 6
Total Area is 6.0
Volume of 10442 TETRA_4 is 0.358048
Volume of 4040 PENTA_6 is 0.590489
Volume of 314 PYRA_5 is 0.051463
Total Volume is 1.0
```

Note

ANSYS ICEM CFD is essentially unitless. The area and volume values reported in the message window are the square and cube, respectively of the length units in ANSYS ICEM CFD. For example, if the length units are in meters, the values represent square meters for area and cubic meters for volume.

The length units can be set in the **Model/Units** DEZ.

Element Info

The **Element Info** option prints the following information for the selected elements in the message window:

- Number of elements selected
- Element type
- Element part
- Node numbers
- Element Thickness (for surface elements)
- Mesh Quality (for mesh elements, if Quality metrics are selected under Settings > Meshing > Quality Info)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If more than one element is selected, the information for each element is printed out in the message window.

Node Info

The **Node Info** option prints the following information for the selected nodes in the message window:

- Number of nodes selected
- Node number
- Node dimension
- Node coordinates
- Thickness at node (for surface elements)
- Mesh Quality (for mesh elements, if Quality metrics are selected under Settings > Meshing > Quality Info)

If more than one node is selected, the information for each node is printed out in the message window.

Element Type/ Property Info

The **Element Type / Property Info** option prints information about element types and defined material properties in the message window.

Toolbox

The **Toolbox** option will open a window in the lower right hand corner of the GUI that includes a calculator, notepad, unit conversion tables, and a variables/parameters window.

Note

Variables may be used instead of numerical values in scripts and geometry creation functions.

Project File

The **Project File** option prints information regarding the project file in the message window.

Domain File

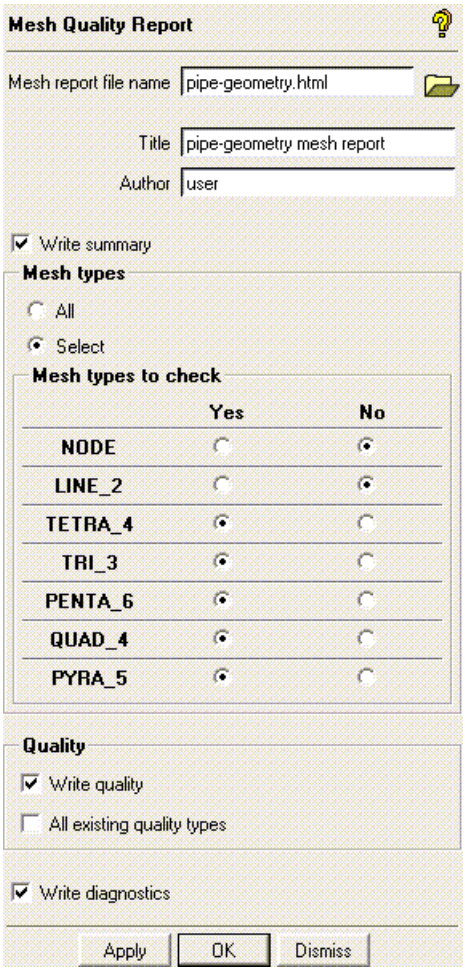
The **Domain File** option prints information about the domain file in the message window.

Mesh Report

The **Mesh Report** option allows you to generate a mesh quality report using the options in the **Mesh Quality Report** DEZ.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 60: Mesh Quality Report DEZ



Mesh report file name
specifies the name of the mesh report file.

Title
specifies title of the mesh report project.

Author
specifies the author of the project.

Write summary
enables the writing of the summary for the mesh.

Mesh types
contains options to report available mesh statistics for all or selected element types.

All
reports available mesh statistics for all available element types.

Select
limits the mesh statistics report to only check the selected element types.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quality

Write quality
enables the writing of the quality for the mesh.

All existing quality types
enables the writing of the quality report for all existing quality types for the mesh. The list of quality measures reported is dependent on the loaded mesh. This option is disabled by default. When disabled (default), the mesh report will include the quality report for only the **Quality**, **Determinant**, **Aspect Ratio**, **Min angle**, **Max ortho**, **Max warp**, **Max warpgls**, and **Skew** measures.

Write diagnostics
enables the writing of the diagnostics for the mesh.

Settings Menu

The **Settings** menu contains different setting options that allow you to change the default settings of display, memory, speed, etc.

Figure 61: Settings Options

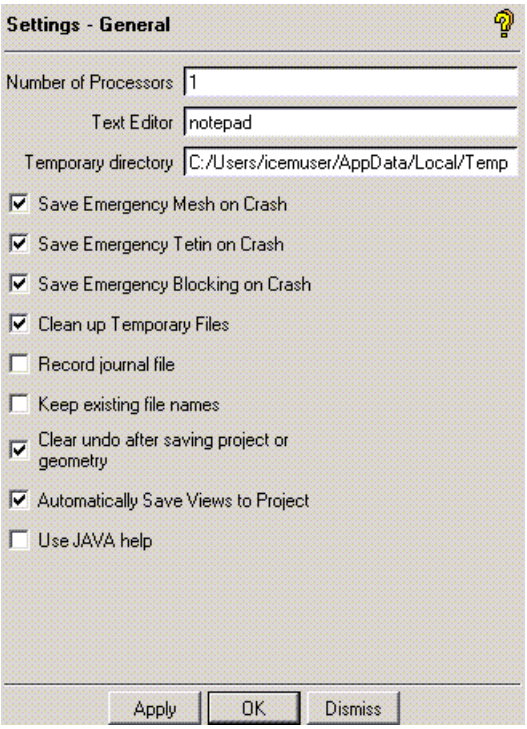
The options available are as follows:
General
Product
Display
Speed
Memory
Lighting
Background Style
Mouse Bindings/Spaceball
Selection
Remote
Model/Units
Geometry Options
Meshing
Solver
Reset

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General

The **General** setting options allow you to change the default settings regarding files, number of processors, text editor, etc., for **ANSYS ICEM CFD** in the **Settings-General** DEZ.

Figure 62: Settings-General DEZ



Number of Processors

specifies the number of processors. Some operations such as tetra meshing, hexa meshing, and smoothing can use multiple processors if they are available. Specifying multiple processors will enhance the performance of these operations. You can specify a value from **0** to **256**. The default is **1**. If the **Number of Processors** is set to **0**, the number of processors will be set automatically to the maximum number of processors available.

Text Editor

defines the default text editor that will be used by **ANSYS ICEM CFD**. For example: **Notepad** for Windows and **vi** for UNIX systems.

Temporary Directory

defines a temporary directory where **ICEM CFD** data files are written. To define the location, list the path of an existing directory using UNIX notation (for example, `c:/users/temp` instead of `c:\users\temp`). If the directory does not exist, ICEM CFD will not create it.

Save Emergency Mesh on Crash

enables the saving of the mesh file in the event of a program crash.

Save Emergency Tetin on Crash

enables the saving of the Tetin file in the event of a program crash.

Save Emergency Blocking on Crash

enables the saving of the blocking file in the event of a program crash.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clean up Temporary Files
enables the cleaning up of temporary files, if any, inside the project directory.

Record Journal File
creates a journal file (*.jrf) of the Tcl/Tk commands representing all the operations in an entire session. This is similar to replay scripts, except that the journal file is saved in the working directory. The journal files can be accessed using the same options used for Replay Scripts.

Note

Journal files are useful for debugging purposes. Technical Support may utilize these files when diagnosing a problem.

Keep existing file names
enables the use of the existing mesh, geometry, and other file names when saving a project instead of creating new file names to match the project name. These file names are stored with the project and loading the project file will automatically load these files too. This feature is useful when working with large files. If this option is disabled (default), saving the project with a new name will create new files with matching names for geometry, mesh, attributes, etc.

Clear undo after saving project or geometry
enables the clearing of the **Undo** history after saving the project or geometry. If this option is disabled, you will be able to undo operations even after the project has been saved.

Automatically Save Views to Project
enables the saved views to be saved with the project file. This option is enabled by default.

Note

You can use the **View > View Control > Edit/Load/Save Views** option (see the View Control (p. 56) section) to edit the views saved with the project file. The **View Control** appears in the bottom-right corner of the screen.

Use JAVA Help
allows you to open the Java file version of the **Help Manual** when selecting **Help > Help Topics**.

Product

The **Product** settings include options for running ANSYS ICEM CFD. The settings control which tabs are visible in the ANSYS ICEM CFD environment.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 63: Product-Selection DEZ

Product Setup

allows you to set the flavor of ANSYS ICEM CFD. This directly affects which tabs are visible in the interface. The tabs for **Geometry**, **Mesh**, and **Edit Mesh** are visible in all cases.

Note

Changes made to this DEZ do not take effect until the next time the software is started.

Default

determines the default capabilities of ANSYS ICEM CFD available for a user based on the license key. If the **`aienv`** and **`aioutcfd`** key are available, the default is **FEA + CFD Utilities**. If only the **`aienv`** key is available (without **`aioutcfd`**) then the **FEA Version** is the default. If an **`aimed`** key (comes with Tetra or Hexa) is drawn along with **`aioutcfd`**, the default will be the **CFD Version**. If the **`aimed`** key is not accompanied by an **`aioutcfd`** key, then **ANSYS solvers – CFD Version** is the default. If you have the **`aiaddon`** key, then **ANSYS Solvers – FEA Version** is the default.

Note

If a user has all of these keys, the default will use the order above. The user can override this by specifically choosing one of the other radio buttons. If a tab is chosen without access to the required keys, the message window will display a license error when the user tries to access that tab.

FEA + CFD Utilities

displays all tabs. This includes access to all CFD solvers, CFD utilities, and advanced setup options for FEA solvers like ANSYS, LS-DYNA, and Nastran.

FEA Version

shows the **Properties**, **Loads**, **Constraints** and **Solve Options** tabs, but not the **Output** (CFD) tab. You can set advanced boundary conditions for FEA solvers like ANSYS, LS-DYNA, and Nastran.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFD Version
shows the **Output** tab, but not the **Properties**, **Loads**, **Constraints** or **Solve Options** tabs. Separate license keys for Quad, Tetra, and Hexa meshers are required.

ANSYS Solvers - CFD Version
similar to the **CFD Version**, except that the **Output** tab shows buttons for ANSYS solvers instead of the larger list of solvers.

ANSYS Solvers - FEA Version
similar to the **FEA Version**, except license limited to only export to ANSYS, Autodyn and LS-DYNA.

Blocking
enables the **Blocking** tab. The ANSYS ICEM CFD Hexa license (**`aihexa`**) is required.

GUI Style
specifies the style for the GUI. You can select either the **Workbench** style or the traditional **ICEM CFD** style. The **Workbench** style has the display tree on the upper-left and the DEZ on the lower-left of the interface. The background colors are set up to match those in ANSYS Workbench and the ANSYS logo is displayed in the upper-right corner of the graphics window. The message window is wider and spans the entire window when there is no histogram, scan, index, or interrupt window present. This is the default setting for ANSYS ICEM CFD. The **ICEM CFD** style has the positions of the display tree and the DEZ swapped, with a solid background color.

Note

This is not the same as the option to use the old GUI.

Beta Options
enables beta options within ANSYS ICEM CFD to be made available. These options are new features that are pending more extensive testing and documentation before being added to the standard release.

Note

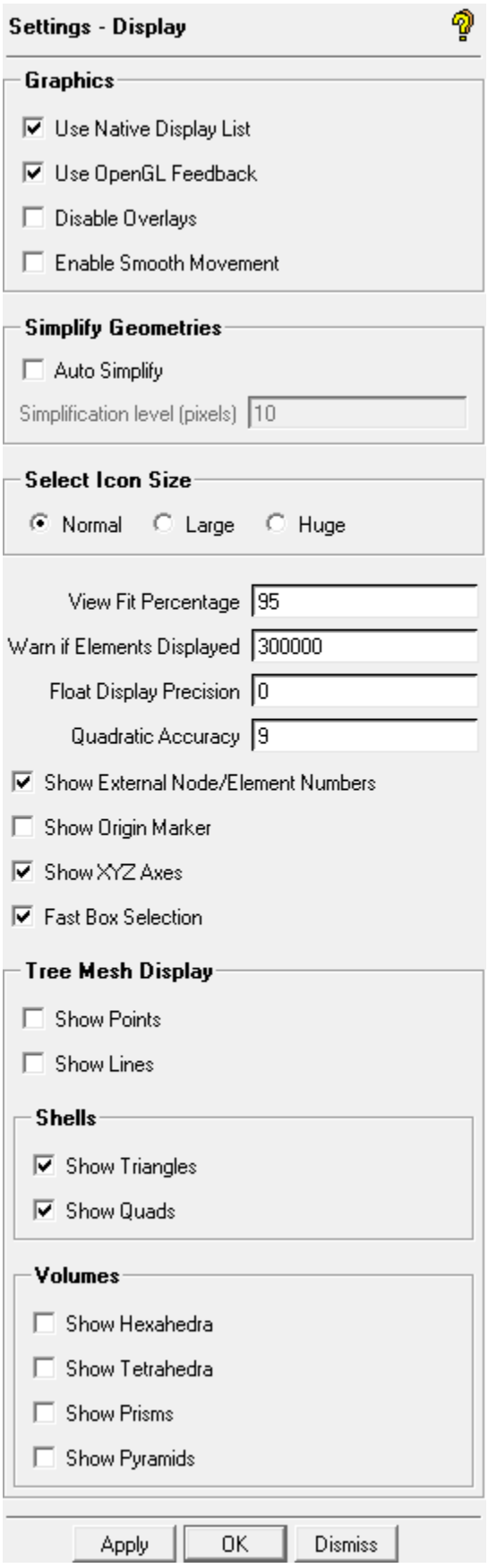
After enabling any option, you must quit and restart the software to see the changes.

Display

The **Display** options allow you to change the display settings for **ANSYS ICEM CFD** using the options in the **Settings-Display** DEZ. Some changes in settings will be effective only after restarting **ANSYS ICEM CFD**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 64: Display Settings window



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Native Display List
enables the use of OpenGL or another accelerated graphics library when possible. The final color including the lighting effects are fixed until the surfaces are refreshed. This option is recommended, because the color and lighting effects are updated for every redraw. Depending on the system and machine, this option may result in slower display redraws. Some nVidia graphics cards may show slower performance when this option is enabled.

Note

If you encounter graphics performance problems, disable the **Use Native Display List** option and click **Apply**. You do not need to restart ANSYS ICEM CFD for this change to take effect.

Use OpenGL Feedback
enables the use of OpenGL's feedback mechanism to detect the 3D coordinates of the location on the model that the mouse is pointing to. This option is recommended when the option **Use Native Display List** is enabled, since it allows the program to release the memory holding the model's geometry data once it has been used to create the display list.

Disable Overlays
disables the use of overlays in drawing a selection box since some graphical cards perform slower with this option.

Enable Smooth Movement
Enable Smooth Movement improves display performance, allowing you to rotate very large models smoothly. With this option enabled, you will need to double click to change the model's center of rotation, in **Dynamic Mode.** See The ANSYS ICEM CFD GUI in the User's Manual and Mouse Bindings/Space-ball (p. 81) in the Help manual.

Simplify Geometries
contains options for simplifying the geometry rendering.

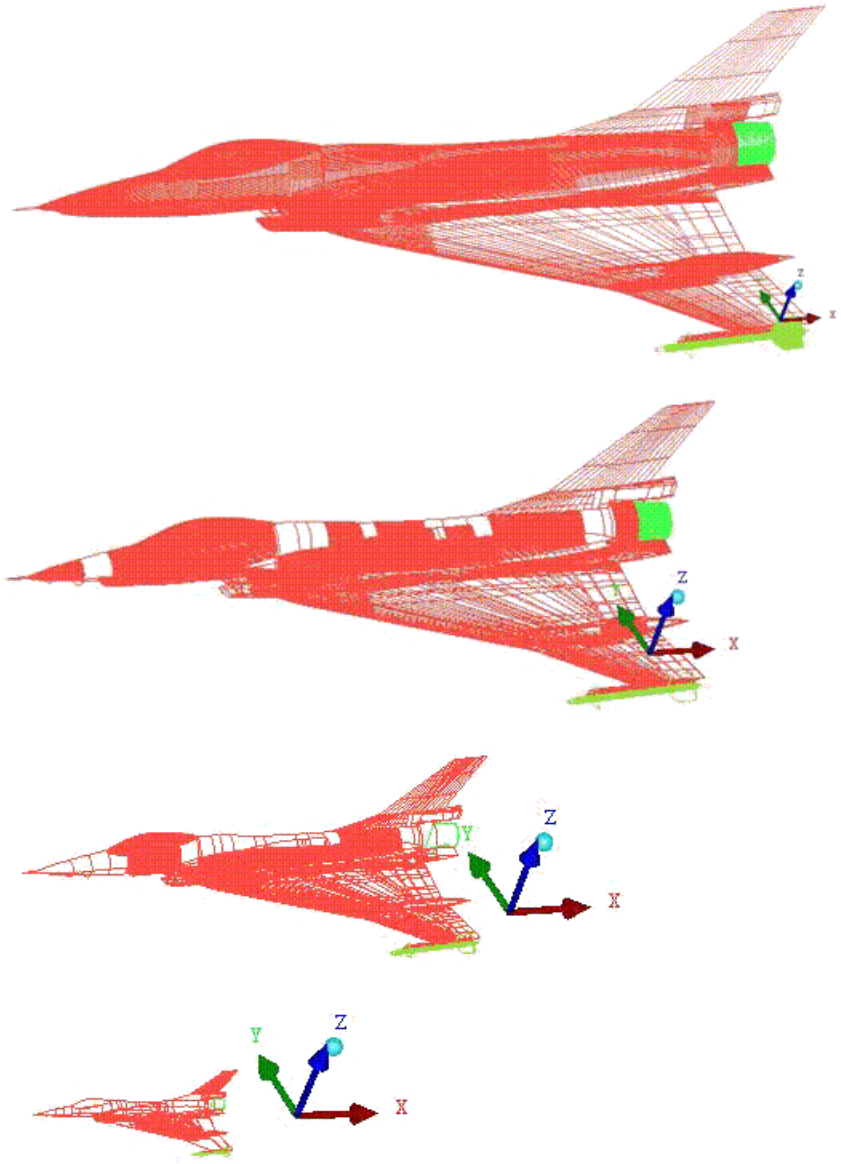
Auto Simplify
simplifies geometry rendering as the model is zoomed out. This option can speed up rendering of large assemblies or models containing a large number of surfaces. The **Simplification level (pixels)** option sets the threshold for simplification.

Simplification level (pixels)
sets the threshold for simplifying geometry rendering. As the model is zoomed out, surfaces of less than the specified number of pixels across are displayed in a more simplified way in order to speed up graphics performance. The default is 10 pixels, meaning that as the model is zoomed out and each surface becomes less than 10 pixels across, its display is simplified.

Figure 65: Simplifying Geometry Rendering (p. 76) shows a jet with surfaces shown as **Wireframe** with **Show Full** enabled, and **Simplification level** set to **50** pixels. As the model is zoomed out, the smaller surfaces fall below the 50 pixel threshold, and the display is simplified.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 65: Simplifying Geometry Rendering



Select Icon Size

specifies the icon size. You can select **Normal**, **Large**, or **Huge**, as required. The software needs to be restarted before any change will take effect. If your screen resolution is very high, you may prefer the **Huge** setting.

	ICON SIZES	
	Tab	**DEZ (Data Entry Zone)**
Normal	24	35
Large	35	35
Huge	35	48

The font size will also change depending on the icon size set and the screen resolution.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

View Fit Percentage

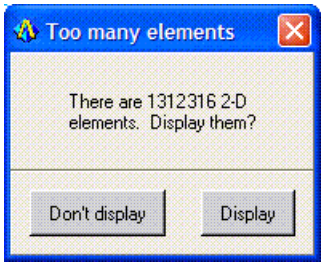
determines the percentage of the screen on which the model will be displayed when the **Fit to Screen** option is used. For example, if set to 90, when Fit to Screen is selected, the model will displayed on 90 percent of the total height and width of the screen display. The default of 95% is usually sufficient to prevent the on-screen prompts from obscuring a portion of the model.

Warn if Elements Displayed

sets the threshold number of elements that will be displayed without a warning. These elements are displayed according to the **Tree Mesh Display** settings after the mesh is loaded. The default value is 1000000, and the value 0 will disable the warning completely.

In the case of Tetra Octree mesh, this warning is displayed before the in-process smoothing. If the specified number is exceeded, a popup window will tell you the number of elements about to be displayed and ask if you want them to be displayed.

Figure 66: Example of Display Elements Window



Note

If you are generating large numbers of shell elements and do not want to display them, disable them under **Tree Mesh Display**.

Float Display Precision

sets the number of decimal places displayed.

Quadratic Accuracy

makes the Bspline display a smoother curve.

Show External Node/Element Numbers

displays the external Node or Element information. Internally, these numbers may be different in order to optimize the meshing routines, but most users prefer to interact with the external numbers as they were before being imported into **ANSYS ICEM CFD** or as they will appear after export. This affects the behavior of node and element number display, as well as features such as creating a subset from node numbers.

Show Origin Marker

displays the origin marker on the screen.

Show XYZ Axes

displays the XYZ axes in the bottom right corner. This triad is interactive and clicking on it will orient the display.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

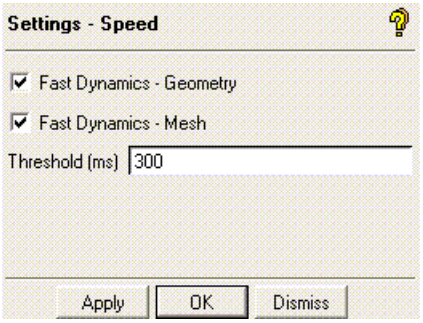
Fast Box Selection
enables the "-overlay" option for the selection box. This can increase performance when there are millions of selected items. Since the memory or accuracy penalty is small, it is recommended that it is enabled for all models.

Tree Mesh Display
allows you to select the default elements which are displayed when a geometry or mesh is loaded. You can enable the display of points, lines, shell elements (triangles and quads), and volume elements (hexahedra, tetrahedra, prisms, and pyramids) as required. On a large model, displaying all the **Shells** by default may take too long, so you may want to disable the shell elements. When computing a large tetra mesh, disabling **Show Triangles** would prevent the time consuming display of these elements before and during the in-process smoothing steps.

Speed

The **Speed** options allow you to modify display modes related to geometry and mesh.

Figure 67: Speed Options window



Fast Dynamics - Geometry
enables a point display mode for the geometry to improve rotational performance.

Fast Dynamics - Mesh
enables a point display mode for the mesh to improve rotational performance.

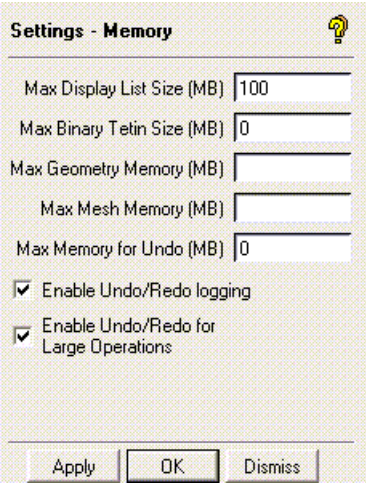
Threshold (ms)
specifies the length of time between the last input and the time when the display refreshes.

Memory

The **Memory** options contain features related to the program's maximum memory parameters.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 68: Memory Related Features



Max Display List Size (MB)
specifies how much memory is dedicated to making redraws faster.

Max Binary Tetin Size (MB)
specifies the maximum amount of memory that a binary Tetin file can take up. A value of 0 means no limit.

Max Geometry Memory (MB)
specifies the amount of memory for use by the geometry. A value of 0 is the recommended value since if this space is exceeded it will not load anything.

Max Mesh Memory (MB)
specifies the amount of memory for use by the mesh. 0 is the recommended value since if this space is exceeded it will not load anything.

Max Memory for Undo (MB)
specifies the maximum amount of memory that the undo option will take up. A value of 0 means no limit.

Enable Undo/Redo logging
specifies whether or not to reserve memory for the availability of undo for log messages.

Enable Undo/Redo for Large Operations
specifies whether or not to reserve memory for the availability of undo for larger operations.

Note

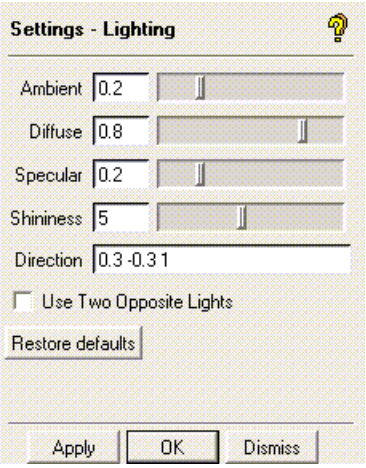
Disabling this option also allows a more efficient algorithm (reduced memory usage) for Linear to Quadratic conversion.

Lighting

The **Lighting** options allow you to interactively set the direction and different components of the lighting on entities displayed in solid view.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 69: Settings-Lighting DEZ



Ambient
specifies the ambient light setting. The Ambient setting has a range of 0 to 1.

Diffuse
specifies the diffuse light setting. The Diffuse setting has a range of 0 to 1.

Specular
specifies the specular setting. The Specular setting has a range of 0 to 1. For more uniform shading, lower the specular component.

Shininess
specifies the shininess setting. The Shininess setting has a range of 1 to 10.

Direction
specifies the direction of the lighting as a vector.

Use Two Opposite Lights
enables the use of two opposite lights. For faceted data, the default lighting settings may not be ideal due to the differing orientations of the surface facets. In such a case, utilizing the **Use Two Opposite Lights** option, and lowering the specular component to zero will remove the contrast and provide more uniform shading.

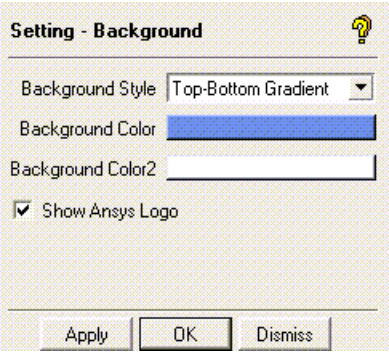
Restore defaults
restores all lighting settings to their default values.

Background Style

The **Background Style** options allow you to select the background style and colors.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 70: Settings-Background DEZ



Background Style
allows you to select a solid background, or one of the following gradients: Top-Bottom, Left-Right, or Diagonal.

Background Color
specifies the background color selected from the menu of colors. Click the palette icon to open a separate window with a color map. To select a color from the colormap, move the cross-hairs to the desired color, and use the sliding arrow on the right to select the desired shade. Click on a blank square under **User colors** to make the specified color an option. Click on the desired color square and click **Apply** to select it as the background color.

Background Color2
specifies the second background color selected to form gradients along with the first Background Color.

Show ANSYS Logo
enables the use of the ANSYS logo in the graphics window.

Mouse Bindings/Spaceball

The default functions for the mouse buttons and space ball are shown in Figure 71: Default Spaceball and Mouse Bindings (p. 82). and Table 3: Default Mouse Bindings (p. 82)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 71: Default Spaceball and Mouse Bindings

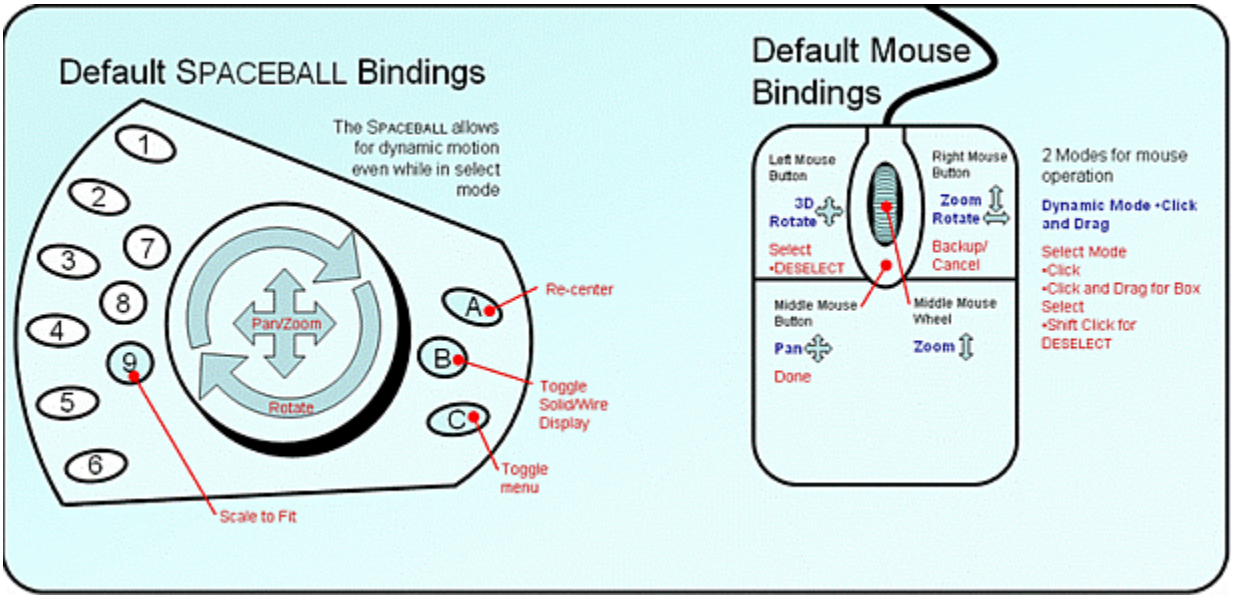


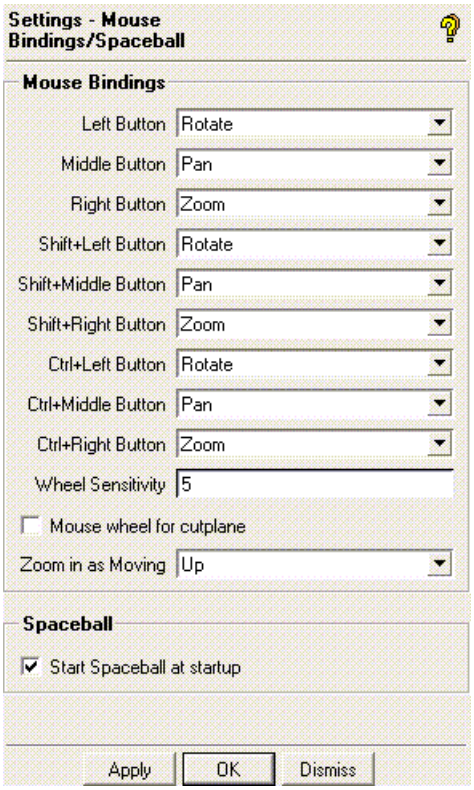
Table 3: Default Mouse Bindings

Button	Dynamic Mode function	Selection Mode function
Left Button	3D Rotate	Select / DESELECT (with Shift Key)
Middle Button	Pan	Done selection
Right Button	Zoom / Rotate	Backup / Cancel
Scroll Wheel	Zoom	

The **Settings>Mouse Bindings/Spaceball** DEZ allows you to redefine the dynamic and selection operations of the mouse buttons.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 72: Settings-Mouse Bindings/Spaceball DEZ



The left, middle, and right mouse buttons all must have different settings.

Wheel Sensitivity
controls the mouse wheel sensitivity.

Mouse wheel for cutplane
enables the manipulation of the cut plane slider with the mouse wheel when a mesh cut plane is enabled. You may disable this option when working with large meshes, so that accidental scrolling does not cause the entire volume mesh to be re-rendered.

Zoom in as Moving
allows you to zoom in as you scroll up or down.

The **Spaceball** option allows you to control the Spaceball check during the ANSYS ICEM CFD startup. A Spaceball (http://www.3dconnexion.com/) is a 3D mouse that lets you Pan, Zoom and Rotate as if you are holding the model in your hand, while you use your traditional mouse with your other hand to Select, Edit, etc. ANSYS ICEM CFD allows simultaneous Dynamic and Selection Mode which can lead to productivity gains, particularly for new users who will find the Spaceball movement to be more intuitive and consistent across tools.

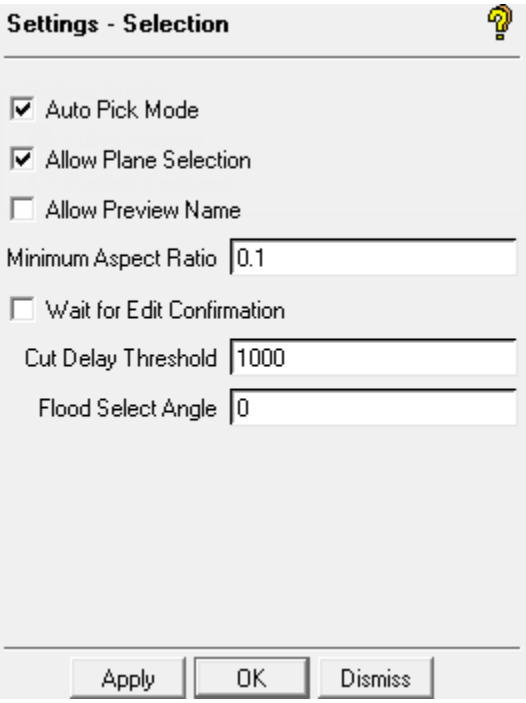
Start Spaceball at startup
enables the check for a Spaceball during startup. This option is enabled by default. If you do not have a Spaceball, disable this option for a quicker startup.

Selection

The **Selection** options allow you to change the default settings related to selection of geometry, mesh, and blocking.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 73: Settings – Selection DEZ



Auto Pick Mode

- initiates the selection process automatically when it is logical to do so.
- activates **continuation mode** for the middle mouse button (behaves like **Apply** plus reiterates the command or proceeds to the next logical selection step.)

Allow Plane Selection

allows the selection tool to snap to planes. When selecting a location on the screen, there is an entity type-based selection hierarchy within a certain proximity of the cursor. The select location tool will snap to a point first, then curve, then surface. If no entity types are within a few pixels of the cursor, and **Allow Plane Selection** is enabled, it will also snap to the X, Y or Z = 0 planes. This option can also be activated during selection using the **Select location** toolbar and enabling **Allow any plane selection**.

Allow Preview Name

enables the displaying of the entity name while previewing selection in geometry selection operations. For example, if in curve select mode, preview selection will highlight the curve as well as the curve name.

Minimum Aspect Ratio

sets the lowest quality that elements are allowed to reach during interactive node movement operations.

Wait for Edit Confirmation

requires the confirmation of selected entities using the middle mouse button after all selections have been made for operations that require a fixed number of selections. If this option is disabled, then the operations will be carried out and finished once all selections are available.

Cut Delay Threshold

specifies the threshold value for updating the cut plane. If the redraw speed takes longer than the specified number of milliseconds, the unstructured cut plane will update only when the mouse is released and not at every stage along its movement.

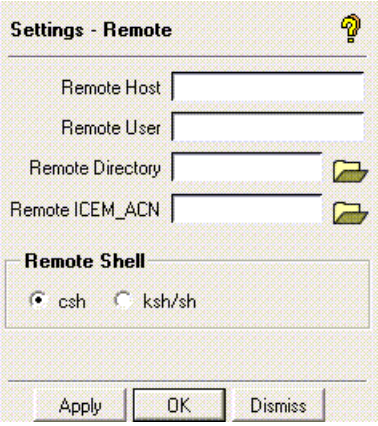
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flood Select Angle
controls the selection of elements. You can change the angle if you want to select neighboring elements with the selection.

Remote

The **Remote** option allows you to run the program on a remote machine. For this, you need to map the remote machine with the proper settings in the **Settings-Remote** DEZ.

Figure 74: Settings—Remote DEZ



Remote Host
specifies the hostname of the remote machine.

Remote User
specifies the user name on the remote machine.

Remote Directory
specifies the directory on the remote machine in which the operation is to be run.

Remote ICEM_ACN
specifies the directory on the remote machine where **ANSYS ICEM CFD** is installed.

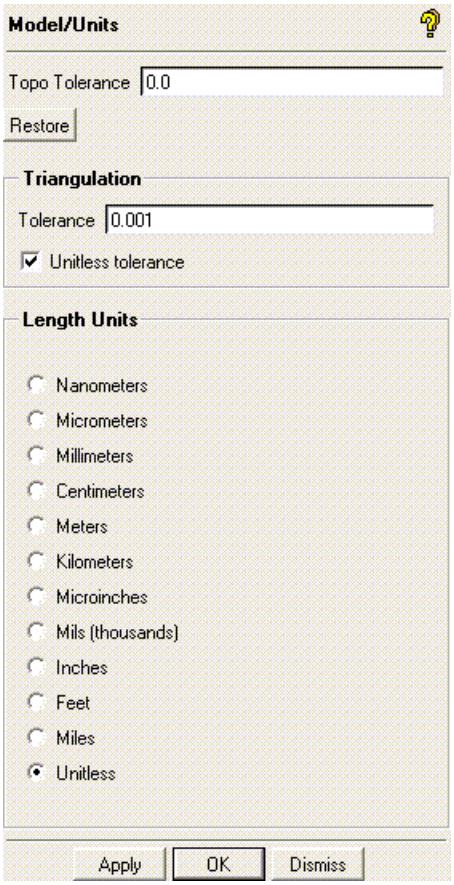
Remote Shell
specifies the type of shell on the remote machine from which the operation will be run.

Model/Units

The **Model/Units** option allows you to set model preferences.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 75: Model/Units Setting Options



Topo Tolerance

specifies the topology tolerance used in operations where topology is automatically calculated.

Restore

resets the topology tolerance to the default value. The default topology tolerance will be set to the smaller of:

- 0.05% of the bounding box diagonal.
- The value so that no more than 5% of the curves in the model are a length less than 2 * the Topo Tolerance value.

Triangulation Tolerance

is the distance allowed between the triangle edge and the actual surface edges. **ANSYS ICEM CFD** triangulates Bspline surfaces and curves when it reads Tetin files. A very low tolerance will give a good representation of the geometry but will require more memory and longer processing time. A higher tolerance will give good performance and less memory usage but a coarser representation of the geometry.

A coarser geometry representation may be a better choice for a large model that will be coarsely meshed. For example, a bigger triangulation tolerance would be reasonable for a full engine assembly model that is going to be shrinkwrapped. If fine precision is required, you may choose a finer triangulation tolerance. For example, some geometry operations, such as trimming a curve with a curved surface will be affected. Also, users who generate very thin boundary layers on curved surfaces may have issues if their surface curvature is not being adequately represented.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

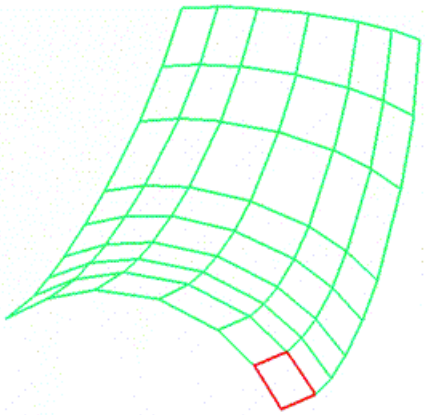
An optimal value for most models is 0.001. This setting is independent of the mesh element size.

Note

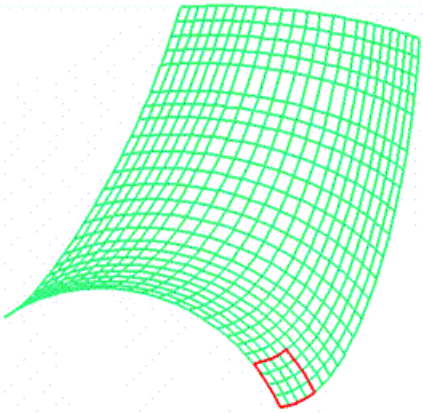
Triangulation tolerance does not affect faceted geometry.

In the examples below, note that for every 1/10th reduction in the Tri Tolerance, each square of the surface representation is further divided into 12-16 squares.

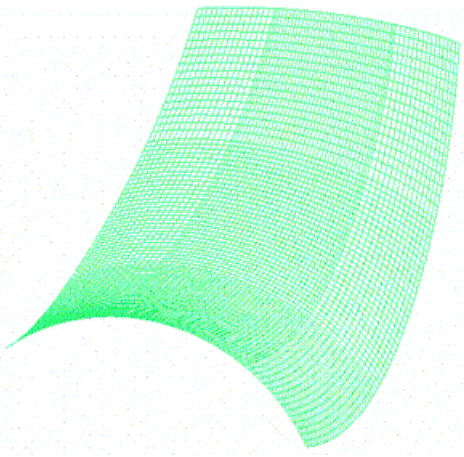
Figure 76: Examples of Triangulation Tolerance



Tri Tolerance = 0.1



Tri Tolerance = 0.01



Tri Tolerance = 0.001 (default)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unitless tolerance

is a factor of the minimum edge length of a given surface or curve. If enabled, the triangulation tolerance is calculated using this factor for each surface. If disabled, the triangulation tolerance is in actual units.

Length Units

allows you to set the length units in ANSYS ICEM CFD. You can select the appropriate unit from the list. The default selection is **Unitless**, where you need to ensure that the intended units are reasonable. In this case, you can use the **Measure Distance** ( ) tool to make sure the model appears to be of the right scale.

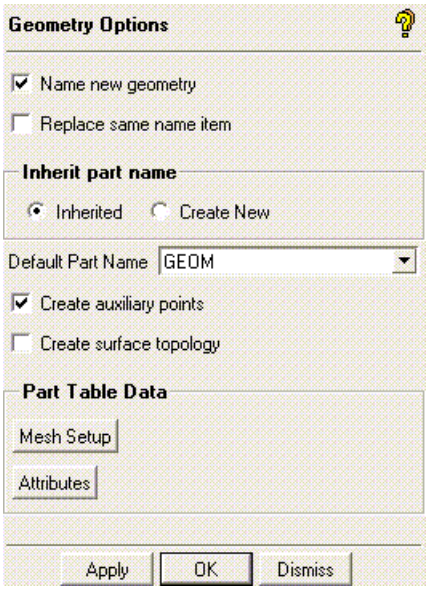
Note

The units information will be saved to the Tetin file.

Geometry Options

The **Geometry Options** allow you to set geometry preferences.

Figure 77: Geometry Options



Name new geometry

enables the display of entity names and modification of entity names when creating geometry entities. If disabled, the entity names will not be visible and the default naming convention will be followed.

Default Naming Convention

If the last two characters of an entity name are digits, then the numbers will be incremented for each new entity created. If the last two characters are not digits, then new entity names will be generated by adding “00” to the name and successively incrementing the number. Examples are shown below.

Entity name	Next Entity Name (Automatically Generated)
abc00	abc01

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

abc.53	abc.54
abc.1	abc.00
abc0.1	abc000

Note

Part names and entity names should be less than 64 alphanumeric characters. Names should start with a letter, not a digit. Evaluators (i.e., "+","-","/", and "*") should not be used in names because they can be misinterpreted as expressions or as denoting an assembly. Part names are written with all upper-case characters. Entity names are case sensitive.

Replace same name item
enables replacing the previously existing entity when an existing name is used for an entity. If disabled, then the new entity will automatically be named according to the default naming convention and the next incremented name will appear in the **Name** field.

Inherit part name
controls the default behavior of new geometry entities that are created.

- **Inherited**

  specifies that all applicable geometry operations will take the part name from the first object selected. This inherited part name will be used for the new objects that are created.

- **Create New**

  specifies that new geometry objects will be created in the working part name (the part name defined in the Point, Curve, Surface, Faceted Data, or Repair forms). Some operations, such as Create Point From Screen, requires a specified working part, so it will take on the working part name even if the Inherited option is used.

Default Part Name
specifies the default working part for any new geometry created in Point, Curve, Surface, Faceted Data, or Repair forms. The working part name for a particular project can be changed independent of this default name.

Create auxiliary points
retains the points used in the creation of geometry.

Create surface topology
creates and maintains the topological connections when geometry is created.

Note

This is currently only used for **Create/Modify Surface** functions.

Part Table Data
These options apply to the **Part Mesh Setup** and **Edit Attributes** tables, with are accessed by right-clicking on the Parts Display Tree.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Setup
opens the **Mesh Setup Data** dialog with the various options available for the Part Mesh Setup table. All options are enabled by default.

Attributes
opens the **Attributes Data** dialog with the various options available for the Edit Attributes table. All options are enabled by default.

Meshing

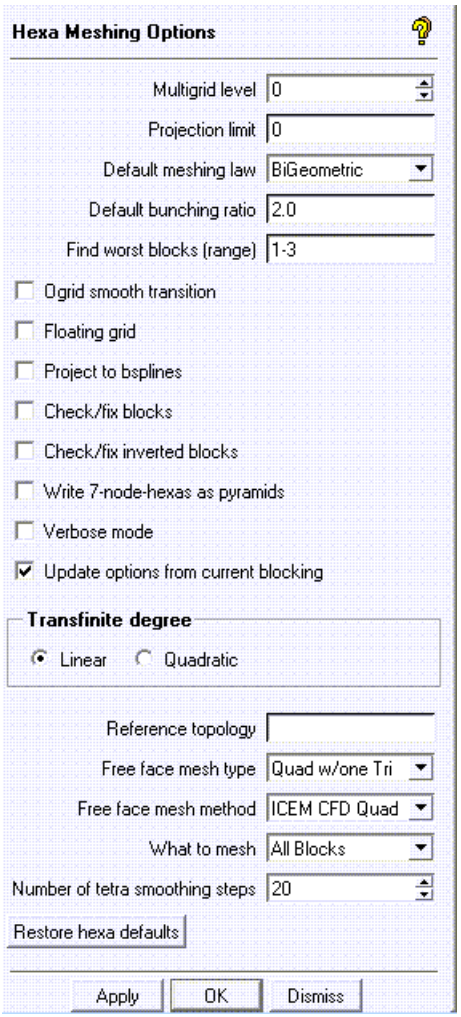
The **Meshing** option allows you to set the preferences for mesh generation. The different meshing setting options are described in the following sections.

Hexa Meshing
Quality/Histogram Info
Edge Info

Hexa Meshing

The **Hexa Meshing** option allows you to specify settings for hexa meshes.

Figure 78: Hexa Meshing Options



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multigrid level

allows you to create a linear multigrid mesh with a specified number of levels. The node count is altered so that the number of elements on an edge is divisible by 2^M, where M is the number of multigrid levels. Thus, the new number of nodes will be:

$$n_{new} = \left(int \left(\frac{(N-1) + 2^M - 1}{2^M} \right) + 1 \right) * 2^M + 1$$

where N is the current number of nodes.

You must select Blocking > Pre-Mesh Params > Update Sizes to for the change to take effect on a blocking with existing node distributions, or set this parameter before prescribing node counts or element sizes.

Projection limit

controls the projection of nodes. If the Projection limit is set to a non-zero value, P, the nodes along an edge within distance P of the end of the edge will not be projected. Instead, they will be linearly interpolated between the first point farther than P and the end. Nodes on the interior of a face within distance P of the edge of the face will not be projected. Instead, they will be linearly interpolated from the interior of the face. If the Projection limit is set to 0, all nodes will be projected.

The Projection limit is set to a non-zero value in cases where you want to keep the nodes on the edges and avoid projection to the underlying surfaces. This option is typically used for Navier-Stokes grids where the grid spacing is small relative to the geometry tolerance. Allowing nodes within a gap in the geometry to project would skew the elements. With a value P set to slightly larger than the gap, the nodes would instead be interpolated. The value may have to be set by trial and error depending on skewness or negative determinants being reported by pre-mesh quality checks.

Default meshing law

specifies the default meshing law used for node distributions for edges. Edges which have a meshing law set will retain their current law unless Blocking > Pre-Mesh Params > Update Sizes is applied.

Default bunching ratio

specifies the default node expansion ratio along an edge from either end of the edge.

Note

Though the settings are saved in the **`.aienv_options`** file, some settings such as the bunching ratio and the meshing law may also be saved in the blocking file. If a blocking file is loaded, it will override the settings saved in the settings file.

Find Worst Blocks (range)

specifies the range of Worst Blocks displayed by the **Find Worst** blocks option.

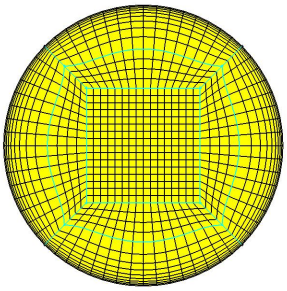
Ogrid Smooth Transition

provides a smooth transition from the offset layer to interior layers. This option uses transfinite interpolation to prevent intermediate unprojected Ogrid splits from adversely affecting the smoothness of the mesh.

In the following figure, the cyan-colored Ogrid edge split, between the outer perimeter and the inner square, is not shaped for optimal mesh quality.

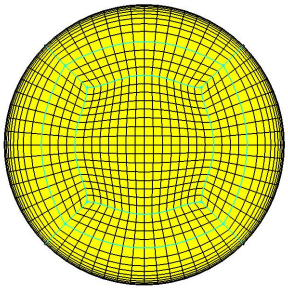
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 79: Ogrid Not Interpolated



This second figure shows the central Hgrid portion and the Ogrid split has been interpolated (shaped) to more smoothly transition the mesh across the model.

Figure 80: Ogrid, interpolated



Floating grid
is used when grid distribution on faces is independent of the sub-edge structure. It also helps to turn on this option if a series of block splits have been made where the splits do not extend through the whole topology. With this option, subsequent splits would not "connect" to previous non-extended splits.

Project to Bsplines
projects the mesh to the true Bspline geometry rather than the faceted representation, which is the internal triangulated representation of surface data as defined by Settings > Model > Triangulation Tolerance. This can be used instead of decreasing the tri-tolerance or using projection limit, where small gaps in the faceted representation create skewed elements on a Navier-Stokes grid. This, however, takes longer and more memory to compute the pre-mesh.

Check/Fix Blocks
checks the internal block data structures for inconsistencies and fixes them if possible.

Check/Fix Inverted Blocks
checks for all the inverted (left-handed) blocks and redefines the block by reordering the vertices to make it right-handed.

Write 7-node-hexas as pyramids
when enabled (by default this option is disabled), converts degenerate 7-noded-hexas (2 nodes merged along an edge) into two pyramids when you select the **Convert to Unstruct Mesh** option under **Blocking > Pre-Mesh** in the Model tree or when using the **File > Blocking > Save Unstructured Mesh** option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Verbose mode

displays additional minor warning messages that are intended to aid development or support in diagnosing possible problems that may occur. This option is disabled by default, but warnings intended to help the **user** with errors or problems are always displayed.

Update options from current blocking

when enabled, this option allows the previously saved hexa settings to be overwritten by the settings from the currently loaded blocking file. When disabled, the previously saved hexa settings will be retained and the settings in the currently loaded file will be ignored. This option is enabled by default.

Transfinite degree

Transfinite interpolation is used for face interpolation when computing a mesh. The default is linear but quadratic is better for maintaining initial element heights. The drawback is that it can produce inverted elements in bowtie-shaped blocks where linear would work fine.

Reference topology

specifies the name of another blocking (the basis for the smoothed mesh) that you have loaded in the same session. If you generate an Euler mesh and the run the mesh smoother on it you can read that mesh back into hexa and create a different blocking that, for instance has Navier Stokes spacing. If the NS blocking references the Euler blocking then it will follow the shape of the smoothed Euler elements to give you a smooth mesh but with the NS spacing.

Unstruct face mesh type

specifies the default mesh type that will be used for meshing unstructured 2D surface blocking. The following options are available:

- Quad w/one Tri
- Tri (STL like)
- All Tri
- All Quad
- Quad Dominant

What to Mesh

specifies the entities or blocks to mesh for **Blocking Tree > Pre-Mesh**. Used typically for blocks with a mixture of mapped (structured), swept, and free (fully unstructured) 3D blocks as encountered using Multizone techniques.

All Blocks

will mesh all blocks. This is the default. Any unstructured (free) blocks will be filled with tetras using the Delaunay algorithm which could take some time for large volumes such as in external flow models.

Vertices

not used.

Edges

will mesh only edges to create line elements.

Faces

will mesh only faces and edges to create surface mesh.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Struct & Swept Blocks
will mesh mapped (structured) and swept blocks, but not free (unstructured) blocks.

> **Note**
>
> This option can be used with multizone blocking to more efficiently iterate and improve the surface mesh and boundary layers. The **All Blocks** option can then be used to mesh the unstructured regions also.

Number of Tetra Smoothing steps
specifies the number of Tetra smoothing steps for Multizone unstructured blocks. You may want to lower this during early iterations of Multizone blocks in order to save time.

Restore Hexa Defaults
restores the hexa meshing settings to the defaults.

Quality/Histogram Info

The **Quality/Histogram Info** option allows you to specify settings for displaying information regarding quality criteria that will be reported for **Element Info** under the **Info** menu. You can also set the default settings for the histogram display that will be used by the Pre-Mesh Quality histogram and the Quality Metric histogram.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 81: Settings-Quality

Check Quality

allows you to select the quality metrics that will be reported for **Element Info** under the Info menu. The default is set to report the **Quality** metric only. For the descriptions of the quality metrics, see Edit Mesh > Display Mesh Quality.

Histogram Defaults

allows you to select the settings for the histogram display.

Note

You can also change the histogram display settings (and ranges) by right-clicking in the histogram window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show
displays the elements within the selected (highlighted) histogram bars.

Solid
displays all selected elements in solid view.

> **Note**
>
> This option may slow down the display speed.

Color by Quality
displays the histogram by colors that corresponds to the quality range. The color contour bar will display the range of quality by color.

Evaluation at
specifies the nodes for which the determinants and distortion will be evaluated for hexa and quad elements. This is used for linear as well as quadratic elements.

- **Corner Nodes**

  The Jacobian determinants of a volume element will be calculated at r = -1, 0, 1; s = -1, 0, 1; t = -1, 0, 1 (natural coordinate system).

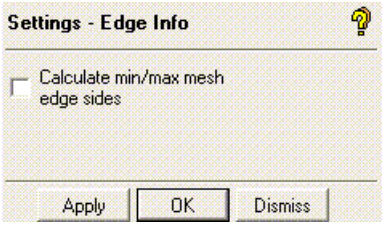
- **Gauss Points**

  The determinants will be calculated at the Gauss points of the selected order.

Edge Info

The **Edge Info** option allows you to specify settings for displaying information regarding edges.

Figure 82: Settings-Edge Info



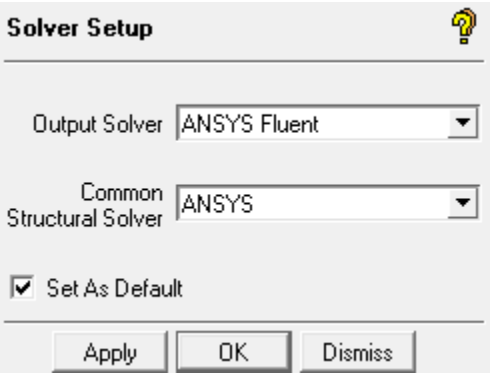
Calculate min / max mesh edge sides
enables the calculation and display of maximum and minimum distances between nodes of a mesh in the information given for Info > Mesh Info.

Solver

The **Solver** option opens a DEZ to select an **Output Solver** (CFD) or **Common Structural Solver** (FEA).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 83: Select Solver DEZ



Output Solver

Select an **Output Solver** from the drop-down list.

More information about the ANSYS ICEM CFD Output Interfaces is available from the **Help** menu. Select the **Output Interfaces** option to open a browser window containing the ANSYS ICEM CFD Output Interfaces information. Select the name of an interface in the Table of Supported Solvers for more detail about that specific interface.

Common Structural Solver

select a **Common Structural Solver** from the drop-down list. The following options are available:

- Nastran
- ANSYS
- LS-DYNA
- Abaqus
- Autodyn

Additional setup for your specific model is done from the **Solve Options** tab.

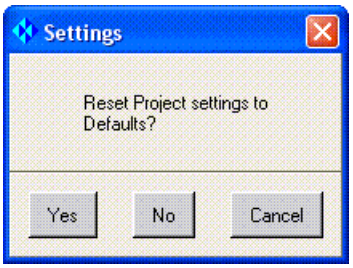
Set As Default

sets the current selection as the default solver.

Reset

The **Reset** option allows you to reset the project settings to the defaults. Click **Yes** in the **Settings** dialog to reset Project settings to the default. This will overwrite the user settings done in the project.

Figure 84: Reset Options Dialog



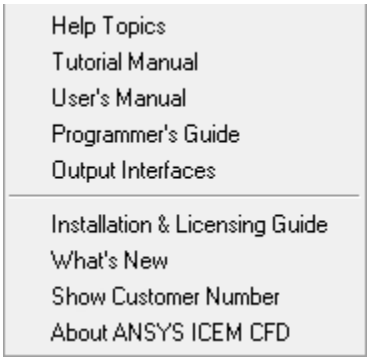
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Help Menu

The **Help** menu provides access to different sections of the ANSYS ICEM CFD Documentation via the ANSYS Help Viewer. On selecting the desired option, the Help Viewer will open at the first page of the selected document.

Help Topics
Tutorial Manual
User's Manual
Programmer's Guide
Output Interfaces
Installation & Licensing Guide
What's New
Show Customer Number
About ANSYS ICEM CFD

Figure 85: Help Options



Help Topics

The **Help Topics** option opens the ANSYS ICEM CFD Help Manual, which contains descriptions of all the features available in ANSYS ICEM CFD.

Tutorial Manual

The **Tutorial Manual** option opens the ANSYS ICEM CFD Tutorial Manual, which contains several text-based tutorial examples.

Additional tutorials and other supporting documents are available on the ANSYS Customer Portal. To access tutorials and their input files on the ANSYS Customer Portal, go to http://support.ansys.com/training. You will require a Show Customer Number (p. 99) to access the Customer Portal

User's Manual

The **User's Manual** option opens the ANSYS ICEM CFD User's Manual, which offers a more technical description of many of the features available in ANSYS ICEM CFD.

Programmer's Guide

The **Programmer's Guide** option takes you to the ANSYS ICEM CFD Programmer's Guide Table of Contents.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD is fully scriptable (in Tcl/Tk) and taking advantage of this is an excellent way to compress your process and use the tools more efficiently. The Programmer's Guide includes:

- ANSYS ICEM CFD scripting basics
- Form creation functions for manipulating the GUI
- External commands for running external programs or performing operations at the OS level
- Scripting commands for use in ANSYS ICEM CFD
- Blocking commands for use in ANSYS ICEM CFD Hexa blocking
- Meshing Directives providing additional scripting commands for use with ANSYS ICEM CFD Meshing, Editing, etc.

Output Interfaces

The **Output Interfaces** option opens the ANSYS ICEM CFD Output Interfaces information in a browser. For information about a specific interface, refer to the Table of Supported Solvers and click the name of the interface.

Installation & Licensing Guide

The **Installation & Licensing Guide** option describes Installation processes and Licensing requirements for ANSYS ICEM CFD.

What's New

The **What's New** option opens the list of additions and improvements to the current release of ANSYS ICEM CFD.

Show Customer Number

The **Show Customer Number** option shows your Customer Number that is included in the license file. The Customer Number is needed to access the Customer Portal.

Note

Reference the Customer Number when contacting ANSYS ICEM CFD Technical Support by phone (1-800-937-3321) or by email (techsupp@ANSYS.com).

About ANSYS ICEM CFD

The **About ANSYS ICEM CFD** option gives information about the version and release date of ANSYS ICEM CFD.

Graphical Main Menu, Utilities and Display Options

Some of the most frequently used functions are made available for quick access in the Figure 86: Graphical Main Menu (p. 100).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 86: Graphical Main Menu

The **Main Menu** icons include options to manage your **Project**, **Geometry**, **Mesh** or **Blocking** files. They function exactly as the equivalent text based menus.

Utilities for taking measurements of your project's features or defining a Local Coordinate system are available along with Undo and Redo.

Display management options include sizing, zooming, and drawing style.

Main Menu Icons
Utilities Icons
Display Management Icons

Main Menu Icons

The **Main Menu** icons include options to manage your **Project**, **Geometry**, **Mesh** or **Blocking** files. They function exactly as the equivalent text based menus.

Figure 87: Main Menu Icons

Open Project

To work on an existing project, select the **Open Project** option.

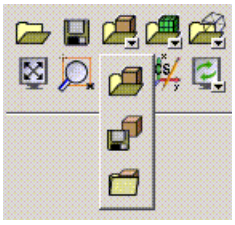
Navigate to the working folder and select the desired project from the **Open Project** window; or select an existing project from the drop down list in the **File name** field.

Save Project

To update the project file (`*.prj`) on your disk, select the **Save Project** option.

This option also updates and saves the data files associated with the project. The files saved are the Geometry file (`*.tin`), Mesh file (`*.uns`), Blocking file (`*.blk`), Parameters file (`*.par`) and Attributes files (`*.fbc` and `*.atr`).

Geometry Menu



The **Geometry** drop-down menu contains options to **Open**, **Save**, and **Close** your Geometry file:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Open Geometry

To load a geometry file into memory and display its graphical image, select the **Open Geometry** icon.

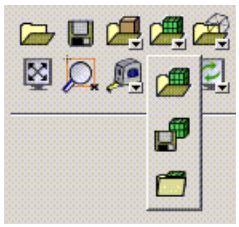
Save Geometry

To update the geometry file with all your recent work, select the **Save Geometry** icon.

Close Geometry

To remove the geometry information from the graphical display and close the loaded geometry file, select the **Close Geometry** icon. If the file has been modified, you will be asked whether you want to save the file.

Mesh Menu



The **Mesh** drop-down menu contains options to **Open**, **Save**, and **Close** your Mesh file:

Open Mesh

To load a mesh file into memory and display its graphical image, select the **Open Mesh** icon.

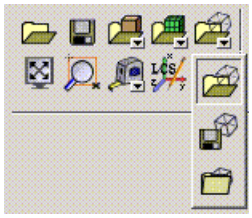
Save Mesh

To update the mesh file with all your recent work, select the **Save Mesh** icon.

Close Mesh

To remove the mesh information from the graphical display and close the loaded mesh file, select the **Close Mesh** icon. If the file has been modified, you will be asked whether you want to save the file.

Blocking Menu



The **Blocking** drop-down menu contains options to **Open**, **Save**, and **Close** your Blocking file:

Open Blocking

To load a blocking file into memory and display its graphical image, select the **Open Blocking** icon.

Save Blocking

To update the blocking file with all your recent work, select the **Save Blocking** icon.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

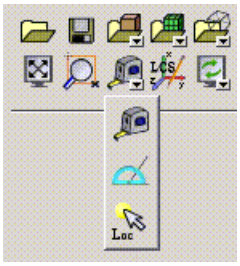
Close Blocking

To remove the blocking information from the graphical display and close the loaded blocking file, select the **Close Blocking** icon. If the file has been modified, you will be asked whether you want to save the file.

Utilities Icons

Utilities for taking measurements of your project's features or defining a Local Coordinate system are available along with Undo and Redo.

Measurement Menu



The **Measurement** drop-down menu contains options to measure **distance** or **angle**, or to find the xyz coordinates of a **location** in the graphical display:

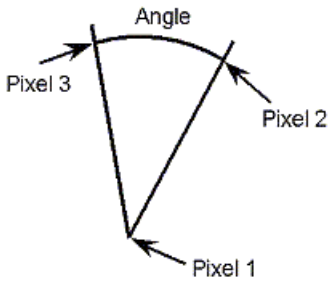
Measure Distance

To measure the distance between two points in the graphical display, use the **Measure Distance** option. Select the locations with the left mouse button. The distance between the two points will be displayed in the message window.

Measure Angle

To measure the angle between two intersecting vectors, use the **Measure Angle** option. Select three points in the graphical display. The first location is at the point of intersection, and the second and third locations define the two different vectors as shown in Figure 88: Pixel Sequence to Measure Angle (p. 102). The angle between these two vectors will be reported in the message window.

Figure 88: Pixel Sequence to Measure Angle



Find Location

To find the xyz coordinates of a point in the graphical display, use the **Find Location** option. Select a pixel with the left mouse button. The location of the point in space will be displayed in the message window and in the GUI.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

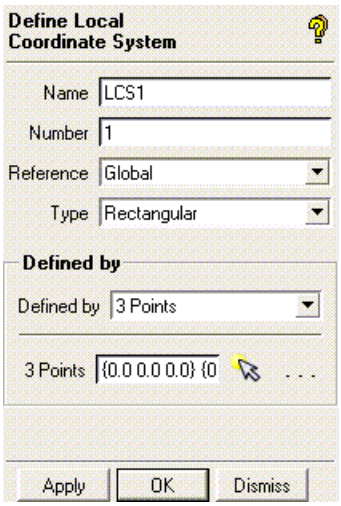
Local Coordinate Systems

You can define local coordinate systems (LCS) for use in geometry, mesh, blocking or boundary condition manipulation. The default coordinate system located at the origin is called **Global**.

Note

LCS information is saved in both geometry (`.tin`) and parameter (`.par`) files. If the geometry is closed, the LCS data will also be closed if only the geometry was loaded. However, if mesh, blocking, boundary conditions, or loads are also loaded, the LCS data will not be removed when the geometry is closed and/or replaced. This will ensure that the LCS info remains in the case that other entities such as mesh, blocking, boundary conditions, or loads are defined in relation to the LCS.

Figure 89: Define Local Coordinate System DEZ



Name
specifies the name for the local coordinate system (LCS). You can define as many systems as you wish.

Number
is used by output interfaces to signify a solver's LCS number.

Reference
specifies the reference coordinate system which is the basis for the new coordinate system.

Type
specifies the coordinate system type. The LCS Type can be Rectangular (Cartesian), Cylindrical, or Spherical. A Rectangular LCS has X, Y, and Z axes. A Cylindrical LCS has R, θ, and Z coordinates. A Spherical LCS has R, θ, and Φ coordinates.

Defined by
specifies the method used to define the LCS. The following methods are available:

Defined by 3 points
In this method, the first point defines the origin of the LCS. The second point defines the positive Z axis relative to the origin for rectangular and cylindrical coordinate systems, and the θ axis (Φ = 0) for spherical coordinate systems. The third point defines positive X for Cartesian coordinate

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

systems, and $\theta = 0$ for cylindrical and spherical coordinate systems. This vector is ortho-normalized if the selected third point is not 90 degrees relative to the first two. Points can be selected by clicking on a graphic location, or by clicking on a prescribed (pre-existing) point.

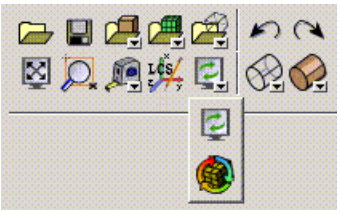
Defined by 1 point

The current screen view plane defines the orientation. The selected point defines the origin. The vector normal out of the screen defines the positive Z axis (Cartesian and cylindrical) or $\Phi = 0$ (spherical). The horizontal direction to the right defines the positive X axis (Cartesian) or $\theta = 0$ (cylindrical and spherical). The vertical direction upward describes the positive Y axis (Cartesian) or $\theta = 90$ (cylindrical and spherical). Points can be selected by clicking on a graphic location, or by clicking on a prescribed (pre-existing) point.

Click **Apply** to create and activate the LCS. A triad showing the coordinate system axes will be displayed in the screen at the point of origin. The Global coordinate system is always displayed in the lower right hand corner (not at the origin). The defined LCS will also appear in the Display Tree.

For more information on options to modify Local Coordinate Systems using the Display Tree, see Display Tree > Local Coordinate Systems.

Refresh / Recompute



The **Refresh/Recompute** drop-down menu contains two options:

Refresh

The **Refresh** option allows you to refresh the screen.

Recompute Premesh

The **Recompute Premesh** option allows you to recompute the mesh.

Undo

The **Undo** option reverses (undoes) the most recently performed operation.

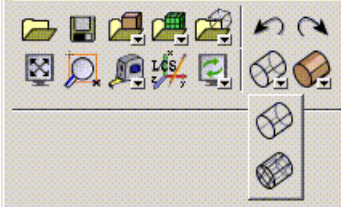
Redo

The **Redo** option repeats the previously reversed operation.

Display Management Icons

Display management options include sizing, zooming, and drawing style.

WireFrame Display Options



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **WireFrame Display Options** drop-down menu contains the following options:

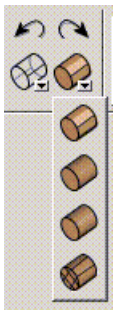
WireFrame Simple Display

The **WireFrame Simple Display** option allows you to display wireframe representation of CAD data or a "Hard Feature" representation of triangulated surface data, including surface boundaries as hard features.

WireFrame Full Display

The **WireFrame Full Display** option allows you to display a more detailed representation of the CAD data, showing more isobars than in the simple representation. For triangulated surface data, a detailed representation will show all the surface triangles.

SolidFrame Display Options



The **SolidFrame Display Options** drop-down menu contains the following options:

Solid Simple Display

The **Solid Simple Display** option allows you to display a smooth or flat shaded representation of surfaces, which is simplified for parametric surfaces.

Solid Full Display

The **Solid Full Display** option allows you to display a smooth shaded representation of surfaces which are triangulated in details.

Solid Full Flat Display

The **Solid Full Flat Display** option allows you to display a flat shaded representation of surfaces which are triangulated in details.

Solid/Wire Full Display

The **Solid/Wire Full Display** option allows you to display a wireframe over a smooth or flat shaded representation of surfaces which are triangulated in details.

Fit Window

The **Fit Window** option scales the model so that it fits in the GUI.

Box Zoom

The **Box Zoom** option prompts you to select a region on GUI to zoom in to.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Entities, Keyboard and Mouse Functions

ANSYS ICEM CFD offers several ways of interacting with your model, including context-sensitive hotkeys and mouse functions.

Selection options, hotkeys, and mouse functions that are appropriate for a given mode (Selection or Dynamic) and Function tab are presented in this chapter.

Selection Options
Hotkeys
Spaceball and Mouse Binding

Selection Options

In ICEM CFD, you may select locations or graphical entities (geometry, mesh, blocking, nodes, and so on) in multiple ways. Common methods are point/click, drawing a surrounding figure, or selecting from a list (name or other identifier). Most selection options are shown in the **Selection Mode Keymap** graphic and supporting table.

Not all selection options are available in all contexts. Available Selection options for each context are presented in a pop-up toolbar in that context. Some selection options appear on the toolbar for only one specific context, although many appear in multiple contexts. Descriptions of the various **Selection toolbars** follow.

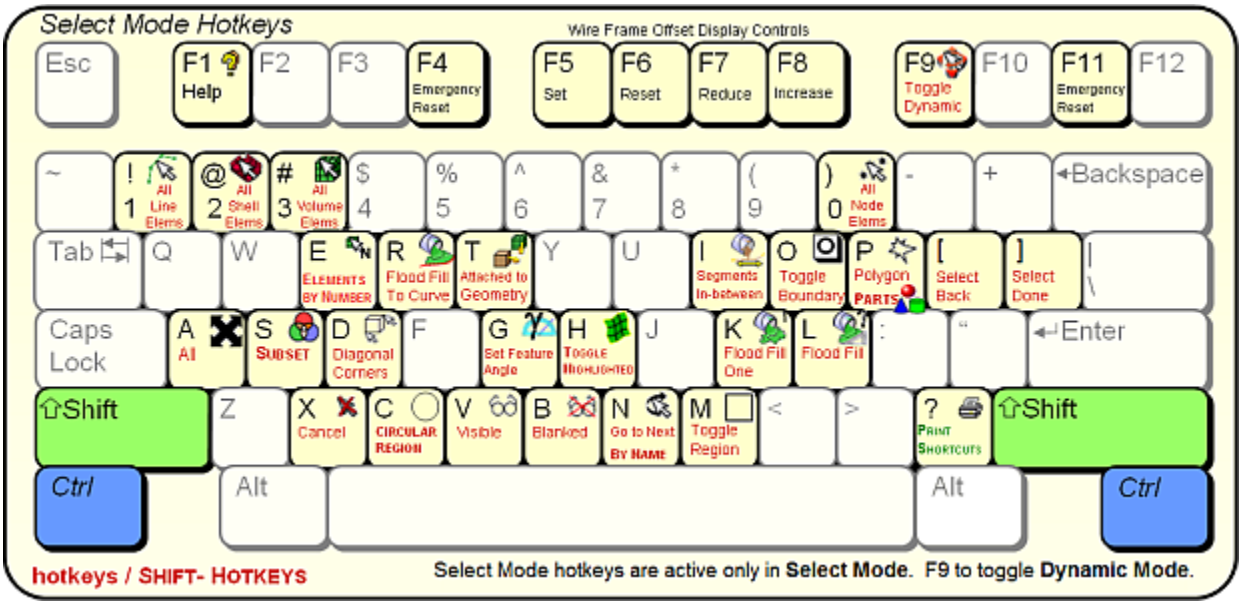
Selection Mode Keymap
Location Selection Toolbar
Geometry Selection Toolbar
Mesh Selection Toolbar
Blocking Selection Toolbars
Density Selection Toolbar

Selection Mode Keymap

Figure 90: Selection Mode Hotkeys (p. 108) and its associated table lists most of the Selection mode functions in ICEM CFD. In any given context, a subset of these will be available, and will be indicated in a pop-up Selection toolbar.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 90: Selection Mode Hotkeys



	Hotkey function	SHIFT + Hotkey function
a	Select all appropriate objects Objects may be visible or blanked Useful for selecting a large number of objects	
b	Select all appropriate blanked objects	
c		**Select items in a circular region** Click (center point) and drag (radius) to define the selection region.
d	Toggle select diagonal corner vertices In certain Blocking selection contexts, this allows you to select multiple, contiguous blocks by clicking two diagonally opposite vertices.	
e		**Select elements by numbers** In certain Mesh selection contexts, this opens a dialog box to specify a range of ID numbers.
g	Set feature angle for flood fill Opens a dialog box to set the maximum angle at which attached items are selected. This applies to	

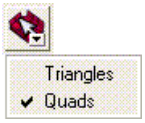
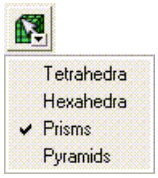
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Hotkey function	SHIFT + Hotkey function
	certain selection functions that allow you to start with one item, and quickly select all attached. If feature angle is set to zero, all attached, appropriate items will be selected.	
h		**Toggle between all edges and highlighted edges** Highlighting of Specific edges may be determined in many ways. For example, selecting Show Single in the Curves Display tree.
i	Select a path in between two selected nodes Preselect two or more, non-contiguous, faceted edges on a faceted surface, and then use this option to select all the edges between them.	**Split two edges and flood fill in between them** This is useful if the edge section you want to select is not clearly delineated into facets.
k	or Select one attached layer Selects items immediately adjacent to the current selection, within the feature angle limit.	
l	or Select all items attached to current selection Selects all attached items, within the feature angle limit.	
m	Toggle between full and partial enclosure Determines whether objects must be fully or partially within the selection area to be included.	**Select items by Material** Used in mesh selection contexts, Materials must be defined and associated with parts.
n	Select the next item picked Cyclically steps through all appropriate objects. Accept with middle mouse button. OR	**Select by numbers** In certain Blocking selection contexts, a dialog box is opened to accept a numeric identifier. OR **Select by name**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Hotkey function	SHIFT + Hotkey function
	Select next edge segment In certain Blocking edge selection contexts, this helps to select a piecewise continuous edge, rather than clicking on individual segments.	In Geometry and Density selection contexts, a dialog box is opened to accept an alphanumeric identifier.
o	Toggle boundary type Active only in (Geometry) Surface or (Blocking) Face selection contexts. Toggles between ON: selecting all surfaces, or faces, regardless of the boundary (inner and outer), and OFF: only the outer boundary surfaces, or faces, will be selected. ON/OFF status is applicable to any surface, or face, selection tool.	
p	Select items in a polygonal region Use the left mouse button to select multiple points until the polygon is created. Middle button completes the selection and closes the polygon. Right button cancels points. After selection, entities may be individually de-selected from the list.	**Select items in a part** A dialog box is opened listing all appropriate parts Check the box to select a part.
r	Select all items attached to current selection, up to a curve In surface mesh selection context, all attached elements within the feature angle limit and constrained by the nearest curve are selected.	**Select elements by Property** Used in mesh selection contexts, Properties must be defined and associated with parts.
s		**Select items in a subset** A dialog box opens listing all appropriate subsets from which to choose.
t	Select mesh attached to geometry Selects the mesh associated with the selected geometry.	

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Hotkey function	SHIFT + Hotkey function
v	Select all appropriate visible objects	**Select block by vertex** Enter vertex numbers in the pop-up dialog box.
x	Cancel selection mode.	
0	Select all node elements	
1	Select all line elements	
2	Select all surface elements A drop down list offers the opportunity to choose which surface element types. 	
3	Select all volume elements A drop down list offers the opportunity to choose which volume element types. 	
[	Undo selection. This is the same as clicking the right mouse button.	
]	Accept selection. This is the same as clicking the middle mouse button.	
F9	Toggle Dynamics Switch to Dynamic mode to reorient the model in the Graphics window, back to Selection mode to complete your selection. The same response is achieved by holding down the Ctrl key.	
?		**Print hotkey list (message window)**

Location Selection Toolbar

When in **Select Location** mode the toolbar shown in Figure 91: Select Location Toolbar (p. 111) pops up showing the **Select location** options.

Figure 91: Select Location Toolbar

The first two icons are selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**

- **Cancel selection**

In addition to the common functions, there are several functions specific to the **Select location** context:

- **Toggle allow any plane selection (key = 1)**

  If toggled ON, a location on any plane can be selected.

- **Toggle allow on XY plane selection (key = 2)**

  If toggled ON, it allows you to select a location on the XY plane.

- **Toggle allow on XZ plane selection (key = 3)**

  If toggled ON, it allows you to select a location on the XZ plane.

- **Toggle allow on YZ plane selection (key = 4)**

  If toggled ON, it allows you to select a location on the YZ plane.

- **Toggle allow on Mesh/Block selection (key = 5)**

  Toggles between allowing you to select a location on Mesh elements or a location on Blocking.

  Note

  It may be easier to select mesh nodes if you disable **Toggle allow any plane selection**.

- **Toggle show location (key = 6)**

  Toggles the display of the selection location coordinates.

- **Toggle show entity name (key = 7)**

  Toggles the display of the selected entity's name.

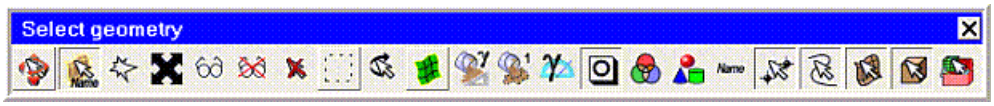
- **Toggle highlight entity (key = 8)**

  Toggles the highlighting of the selected entity.

Geometry Selection Toolbar

When in **Select geometry** mode the toolbar shown in Figure 92: Select Geometry Toolbar (p. 112) pops up showing the **Select geometry** options.

Figure 92: Select Geometry Toolbar



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Many of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select all appropriate objects**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select the next item picked**
- **Toggle between all edges and highlighted edges**
- **Select all items attached to current selection**
- **Select one attached layer**
- **Set feature angle for flood fill**
- **Toggle boundary type**
- **Select items in a subset**
- **Select items in a part**
- **Select by name**

In addition to the common functions, there are several functions specific to the **Select geometry** context:

- **Toggle preselect highlight and name**

  Allows geometry entities to be highlighted and their names to display before you select them. **Enable Preview Name** must be enabled in the **Selection** menu to see the name.

- **Toggle selection of points**

  If toggled ON, points can be selected.

- **Toggle selection of curves**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If toggled ON, curves can be selected.

- **Toggle selection of surfaces**

  If toggled ON, surfaces can be selected.

- **Toggle selection of bodies**

  If toggled ON, bodies can be selected.

- **Toggle selection of mesh**

  If toggled ON, the mesh selection toolbar will appear and mesh elements can be selected.

Segment Selection Toolbar

The **Select segments** toolbar appears when the context requires selection of faceted edges. The **Select segments** options are described below.

Figure 93: Select Segments Toolbar

All of the icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select all items attached to current selection**
- **Select one attached layer**
- **Select a path in between two selected nodes**
- **Split two edges and flood fill in between them**
- **Set feature angle for flood fill**

Mesh Selection Toolbar

When in **Select mesh elements** mode the toolbar shown in Figure 94: Select Mesh Toolbar (p. 115) pops up showing the **Select mesh elements** options.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 94: Select Mesh Toolbar

All except the last of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate objects**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select all items attached to current selection**
- **Select all items attached to current selection, up to a curve**
- **Select one attached layer**
- **Set feature angle for flood fill**
- **Select items in a subset**
- **Select items in a part**
- **Select elements by numbers**
- **Select items by Material**
- **Select items by Property**
- **Select mesh attached to geometry**
- **Select all node elements**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select all line elements**
- **Select all surface elements**
- **Select all volume elements**

In addition to the common functions, the last icon invokes a function specific to the **Select mesh elements** context:

- **Toggle selection of geometry**

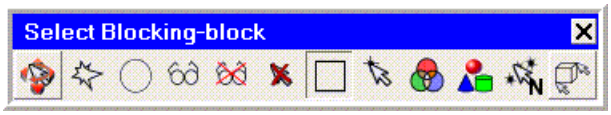
  If toggled ON, the geometry selection toolbar will appear and geometry entities can be selected. This option is available when geometry or mesh selection would be applicable.

Blocking Selection Toolbars

When selecting blocking, you may be required to select **Blocks**, **Faces**, **Edges**, **Compcurves**, or **Vertices**. Each of these has its own toolbar with its specific selection options.

Select Blocking - block

Figure 95: Select Blocking Block Toolbar



All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select block by vertex**
- **Select items in a subset**
- **Select items in a part**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select by numbers**
- **Toggle select diagonal corner vertices**

Select Blocking - face

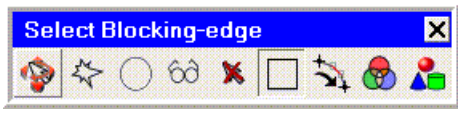
Figure 96: Select Blocking Face Toolbar

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Toggle between all faces and boundary faces**
- **Select items in a subset**
- **Select items in a part**
- **Toggle select diagonal corner vertices**

Select Blocking - edge

Figure 97: Select Blocking Edge Toolbar



All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select next edge segment**
- **Select items in a subset**
- **Select items in a part**

Select Blocking - compcurve

Figure 98: Select Blocking Compcurve Toolbar

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select items in a subset**
- **Select items in a part**
- **Select by numbers**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Blocking - vertex

Figure 99: Select Blocking Vertex Toolbar

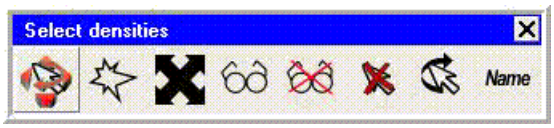
All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**
- **Select items in a polygonal region**
- **Select items in a circular region**
- **Select all appropriate visible objects**
- **Cancel selection**
- **Toggle between full and partial enclosure**
- **Select items in a subset**
- **Select items in a part**
- **Select by numbers**

Density Selection Toolbar

When working with Density regions, the toolbar shown in Figure 100: Select Densities Toolbar (p. 119) pops up showing the **Select densities** options.

Figure 100: Select Densities Toolbar



Note

The **Select densities** toolbar has fewer options than other entity selections. For example, the Enclosure toggle ( ) is not available because only the partial enclosure mode is available.

All of these icons invoke selection mode functions as described in Selection Mode Keymap (p. 107):

- **Toggle Dynamics**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Select items in a polygonal region**
- **Select all appropriate objects**
- **Select all appropriate visible objects**
- **Select all appropriate blanked objects**
- **Cancel selection**
- **Select the next item picked**
- **Select by name**

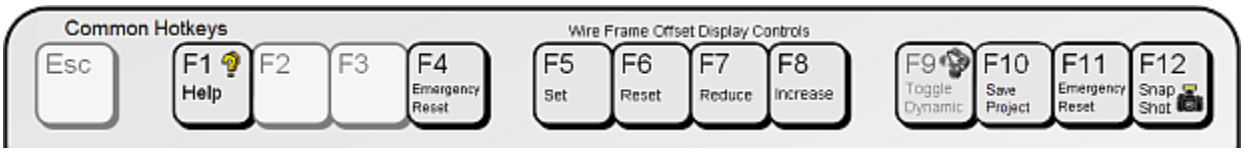
Hotkeys

Many of the most frequently used commands in ANSYS ICEM CFD are accessible via single or combination keystrokes. Many of these hotkeys are common across all contexts while others are specific to the active function group – Geometry, Blocking, or Edit Mesh. To access any hotkeys related to geometry, mesh, blocking operations, you must first go to the respective tab.

> **Note**
>
> Make sure that the Caps Lock is not toggled ON when using hotkeys.
>
> Hotkeys work only when the cursor is in the graphics display area.

Hotkeys which are available in any context are shown in the following image and table.

Figure 101: Common Hotkeys



F1	Open online help
F4	Emergency Reset

> **Note**
>
> Emergency Reset is useful if a series of random keystrokes accidentally locks the keyboard. If a user clicks too fast or bumps multiple keys at a time the keyboard may lock up. The Emergency reset attempts to fix this problem and unlock the keyboard.

F5	Set wireframe offset
F6	Set wireframe offset to 0

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

F7	Increment wireframe offset
F8	Decrement wireframe offset

Note

The Wireframe Offset Display controls allow you to optimize the wireframe display for solid and wireframe modes. This can be useful when the wireframe display is difficult to see in contrast to the solid display. For example, if the wireframe offset is set too high you can see overlapping lines, if it is set too low, the wireframe lines look dim. This can happen when adjusting the zoom, or if the model dimensions make the wireframe offset difficult to compute automatically. When generating images of the mesh it is recommended to increase the wireframe offset slightly to ensure better crispness of solid/wireframe mesh display. Increasing the wireframe display makes the wireframe lines larger and clearer, but it can also bring lines in the background to the foreground. The default Wireframe offset (Z-offset) tries to automatically set the wireframe value based on the model dimensions.

F10	Save Project
F11	Emergency Reset
F12	Save hardcopy
Enter	Apply

Note

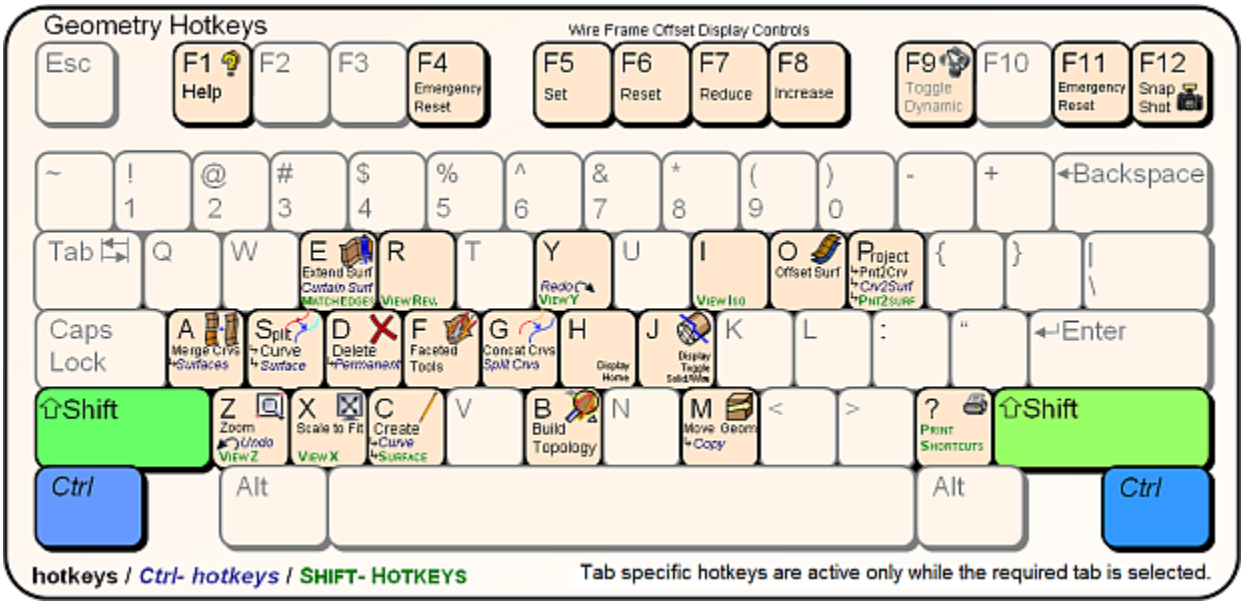
Pressing the Enter key after typing in a field in the GUI is the same as clicking on the Apply button.

Geometry

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Geometry tab functions.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 102: Geometry Hotkeys



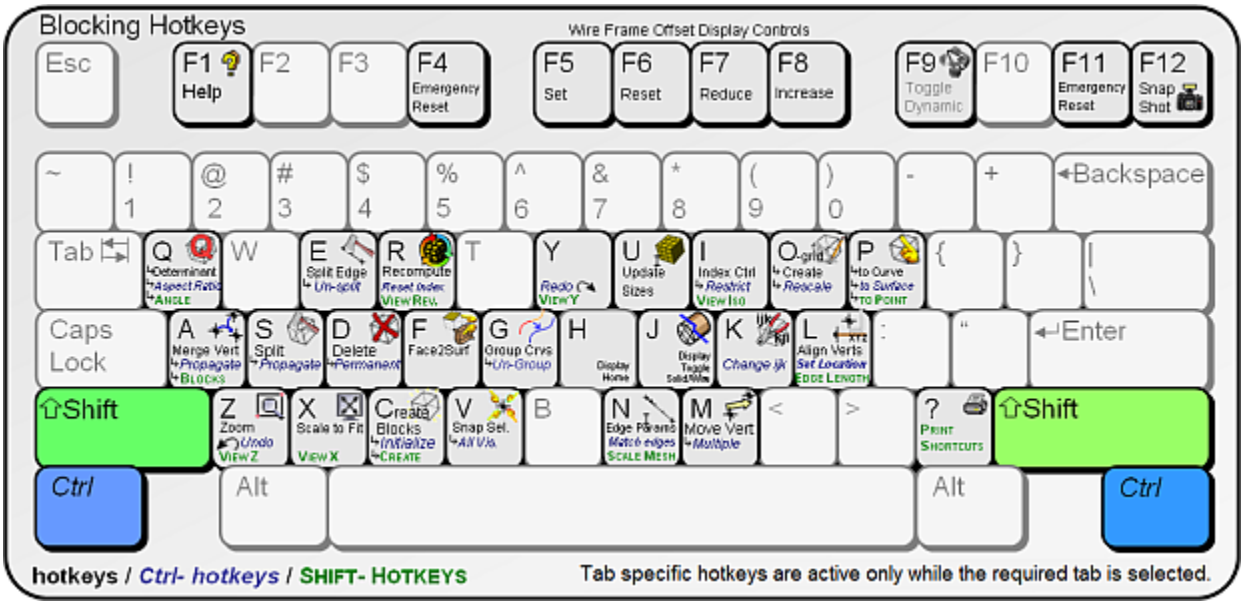
	Hotkey function	*Ctrl* + *Hotkey* function	SHIFT + Hotkey function
a	Concatenate curves	*Merge surfaces*	
b	Build diagnostic topology		
c		*Create curve*	**Create surface**
d	Delete geometry entities	*Delete geometry permanently*	
e	Extend surface	*Create curtain surface*	**Stitch/match edges**
f	Faceted tools	*Move facets to new*	
g	Concatenate curves	*Split curve*	
h	Home position		
i			**Isometric view**
j	Toggle solid display		
m	Move geometry	*Copy geometry*	
o	Offset surface		
p	Project point to curve	*Project curve to surface*	**Project point to surface**
r			**Reverse view**
s	Split curve	*Extend split*	
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blocking

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Blocking tab functions.

Figure 103: Blocking Hotkeys


Blocking Hotkeys
Wire Frame Offset Display Controls
hotkeys / Ctrl- hotkeys / SHIFT- HOTKEYS
Tab specific hotkeys are active only while the required tab is selected.

	Hotkey function	*Ctrl* + *Hotkey* function	SHIFT + Hotkey function
a	Merge vertices	*Merge vertices with propogation*	**Merge blocks**
c		*Initialize blocks*	**Create blocks from vertices**
d	Delete blocks	*Delete blocks permanently*	
e	Split edge	*Unsplit edge*	
f	Associate face to surface	*Fix inverted blocks*	
g	Group curves	*Ungroup curves*	
h	Home position		
i	Index control	*Restrict blocks – corners*	**Isometric view**
j	Toggle solid display		
k		*Change block IJK*	
l	Align vertices	*Set vertex location*	**Set edge length**
m	Move vertex	*Move multiple vertices*	
n	Set edge mesh parameters	*Match edges for node distribution*	**Scale sizes**
o	Create O-grid	*Rescale O-grid*	
p	Project edge to curve	*Project edge to surface*	**Project vertex to point**
q	Check Quality – determinant	*Check Quality – aspect ratio*	**Check Quality – angle**
r	Recompute pre-mesh	*Reset index control*	**Reverse view**
s	Split block	*Extend split*	

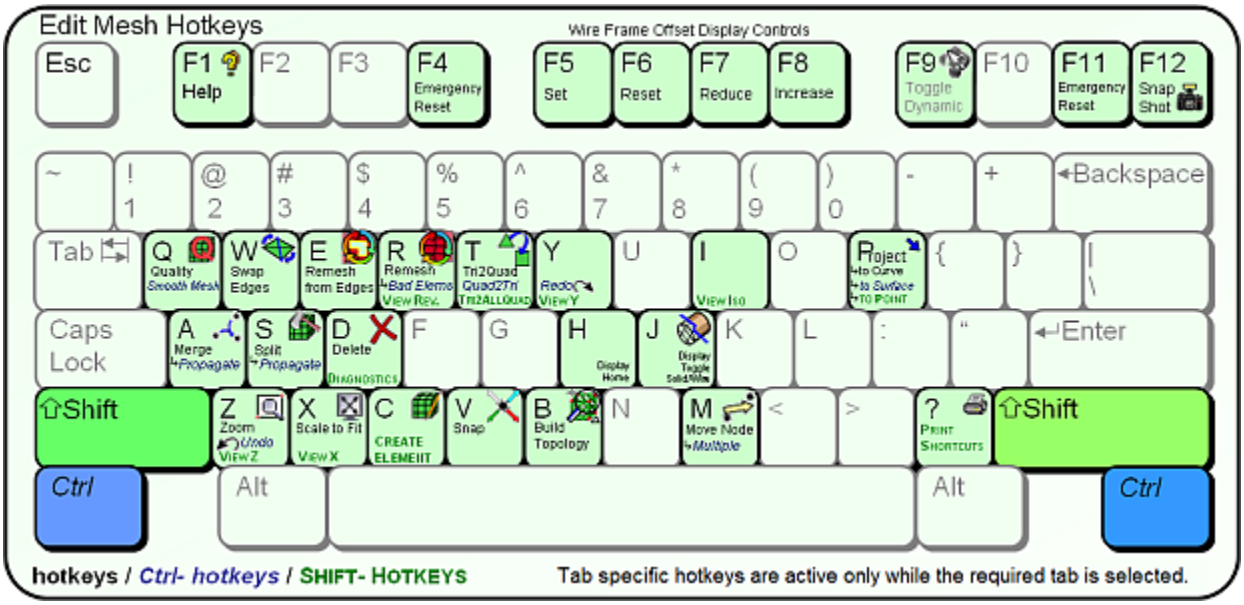
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Hotkey function	*Ctrl* + *Hotkey* function	SHIFT + Hotkey function
u	Update Mesh sizes		
v	Snap selected vertices	*Snap visible vertices*	
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

Edit Mesh

In addition to the common hotkeys listed above, the hotkeys shown in the following image and table are available only when working with the Edit Mesh tab functions.

Figure 104: Edit Mesh Hotkeys



	Hotkey function	*Ctrl* + *Hotkey* function	SHIFT + Hotkey function
a	Merge nodes without propogation	*Merge nodes with propogation*	
b	Build mesh topology		
c			**Create element**
d	Delete mesh elements		**Mesh diagnostics**
e	Mesh from edges		
h	Home position		
i			**Isometric view**
j	Toggle solid display		

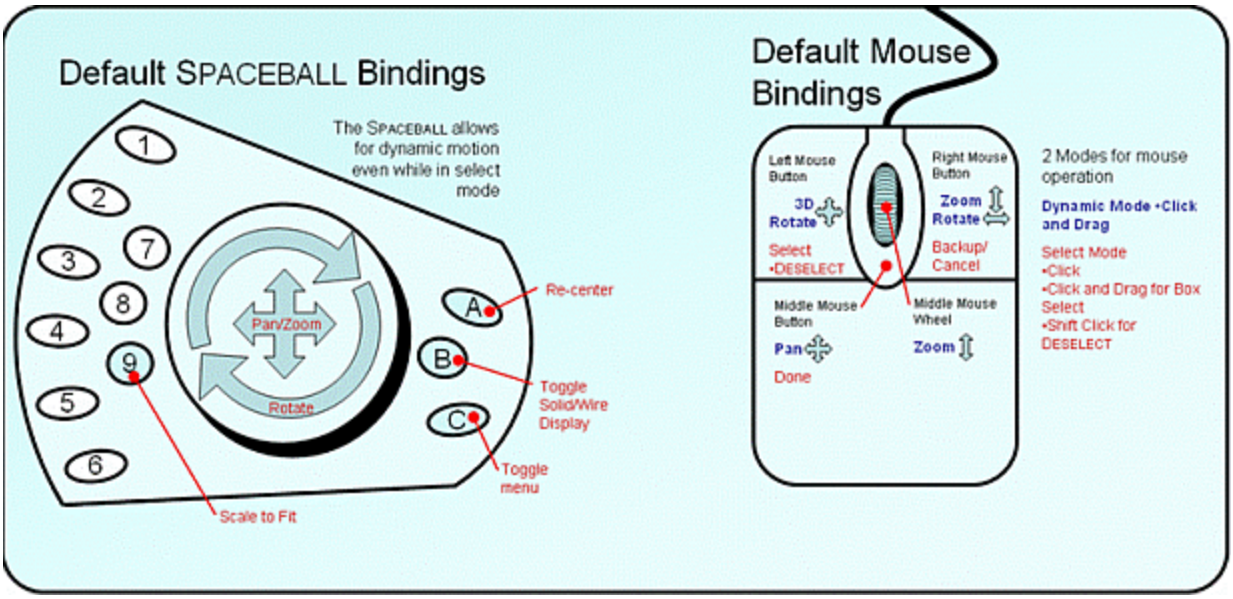
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Hotkey function	***Ctrl* + *Hotkey* function**	**SHIFT + Hotkey function**
m	Move node	*Move multiple nodes*	
p	Project node to curve	*Project node to surface*	**Project node to point**
q	Quality metrics, custom quality	*Smooth mesh globally*	
r	Remesh elements	*Remesh bad elements*	**Reverse view**
s	Split edges without propogation	*Split edges with propogation*	
t	Convert Tri to Quad, quadrization OFF	*Convert Quad to Tri*	**Convert Tri to Quad, quadrization on (refine)**
v	Snap by current projection		
w	Swap edges		
x	Scale to fit		**View in x-direction (right side)**
y		*Redo last undone operation*	**View in y-direction (top view)**
z	Zoom in	*Undo last operation*	**View in z-direction (front view)**
?			**Print hotkey list (message window)**

Spaceball and Mouse Binding

ANSYS ICEM CFD allows simultaneous connection of both a Spaceball and a 3-button mouse. The default actions for these is shown in Figure 105: Default Spaceball and Mouse Bindings (p. 125). For additional information on managing your spaceball and mouse, see Mouse Bindings/Spaceball.

Figure 105: Default Spaceball and Mouse Bindings



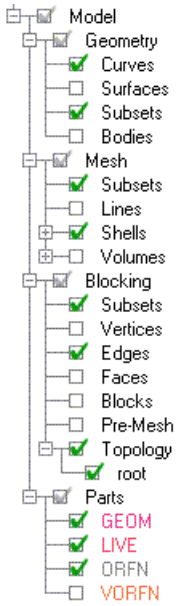
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Tree

Use the **Display Tree**, which is located on the left of the GUI window, to manage what is displayed, and how it is displayed, in the graphics window. **ANSYS ICEM CFD** offers a great deal of control in the mesh generation process. The ability to manage the mesh generation is enhanced if you become familiar with the functions within the Display Tree.

The Display Tree has branches for the four most common types of entities that exist in most projects: Geometry, Mesh, Blocking, and Parts/Subsets. Other branches will appear as various properties, loads, constraints, etc. are applied to the model.

Figure 106: Display Tree



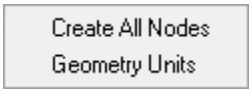
Mouse Usage

Mouse Operation	Description
Right button	Displays Options menu of the selected Display Tree item.
Shift + Left button	To automatically select all the items listed in between two selected (highlighted) items.
Control + Left button	To select items in addition to, or deselect items from, a set of previously selected (highlighted) items.
Left button on check box	Toggles entity visibility.
Double click on Display Tree item with check box	Toggles entity visibility.
Double click on Display Tree item without check box	Opens the Modify options of the selected item.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model

Right-click on **Model** in the Display tree to see the Model display options available, as shown below.



Create All Nodes
Displays all branches available in the Display tree. The default behavior is to display a branch only if entities of that type exist in the model.

Geometry Units
Opens a DEZ in which you can specify a global unit of length to be used in your model.

Choose the appropriate units, and then click **Apply** (DEZ is preserved) or **OK** (DEZ is closed).

Geometry

Click the right mouse button on **Geometry** in the Display tree to see the Geometry display options available, as shown below. The geometry options work only if a Geometry file is loaded.

Figure 107: Geometry Tree Display Options

Show All
Displays the whole geometry.

Hide All
Hides the whole geometry.

Blank Entities
Makes selected entities invisible.

Unblank All Entities
Restores blanked entities.

Rename Entity
Opens a **Rename Geometry** DEZ where you assign a new name to the selected entity.

New Name
Enter the desired name.

Type
Choose the type of geometry from the drop-down list to constrain the selection options.

POINT/CURVE/SURFACE/DENSITY/ENTITY
Click the **Select ...** icon, and then choose an appropriate geometry from the GUI window.

The following sections describe the subparts of the Geometry Tree.
Geometry Subsets Options

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Geometry Subset Options
Geometry Points Options
Geometry Curves Options
Geometry Surfaces Options
Geometry Bodies Options
Geometry Densities Options

Geometry Subsets Options

Elements can be grouped into **Subsets**, which have display options that can be controlled independently.

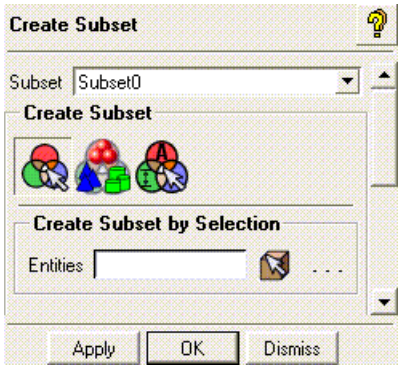
Right click on **Subsets** under **Geometry** to view the display options as shown below.

Figure 108: Subsets Display Options

Create

Creates subsets by the following methods.

Figure 109: Create Subset by Selection Window



Create Subset by Selection

Select geometry or mesh entities to add to the subset, then click Apply. If there are no valid selections and Apply is clicked, then an empty subset will be created.

Note

When entities are selected, the type of entity followed by the entity name appears in the field. For example, the selection of a surface of the entity “box.00” will appear as “surface box.00”. When manually entering an entity, the same format must be used.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Break Part into Subsets by Surface Connectivity

Creates subsets of connected surfaces filtered by the angle of connectivity or surface curvature.

Note

Build Topology must be completed before using this feature.

This feature starts with one of the selected surfaces and returns the connected surfaces that are connected at an angle less than the defined value, or has a curvature less than the defined value. Then it moves on to the next surface. If none of the attached surfaces meet the defined criteria it is placed in its own group.

Surfaces
Select the surfaces to be broken into subsets.

Angle
If the angle between the connected surfaces is less than the defined angle, then it will be added to the same group. An angle of 180 would mean that all connected surfaces would be in the same group. An angle of 0 would mean that all angles would stop surface grouping, so each surface would be in its own group. Parallel surfaces meet at an angle of 0.

Curvature
If the curvature of a surface is less than the defined curvature value, than it will be added to the group. A curvature value of 360 means that no surfaces would be excluded based on curvature, and a value of 0 means that all surfaces would become their own group.

If the default values of angle = 180 and curvature = 360 are used, the selected surfaces will be broken into groups by connectivity. After the groups are created, the subsets are ordered by surface area, where the largest groups are listed first.

Create Subset by Attributes

Create subsets of geometry or mesh entities filtered by specified attributes. Toggle ON the desired type of attribute, and enter the criteria in the field. More than one criteria can be used, for example, entities with mesh maximum sizes between 50 and 100 can be defined by entering "< 100 > 50".

Mesh Maximum Size
To create a subset of surfaces and curves filtered by the Maximum element size set for the surfaces or curves.

Curve Length
To create a subset of curves filtered by curve length.

Surface Area
To create a subset of surfaces filtered by surface area.

Preview Subset
Click to highlight the entities that have the defined attributes. To create the subset of these entities, click Apply.

Show All
Displays all created subsets.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hide All
Hides all created subsets.

Create Part
Creates parts from selected subsets.

> **Note**
>
> Subsets should not overlap since parts are "exclusive groups".

Modify Geometry Subset Options

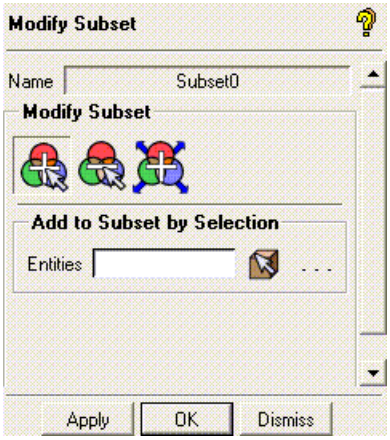
Right click on an existing subset name and the following options will be shown.

Figure 110: Modify Subset Options

Info
Gives information about the subset.

Modify
You can modify Subsets with the following options.

Figure 111: Add to Subset by Selection Window



Add to Subset by Selection

Select the geometric entities to add to the subset.

Remove from Subset by Selection

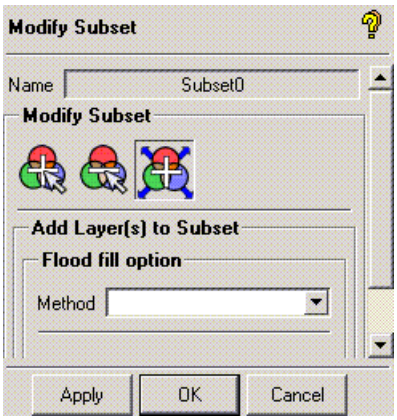
Select the geometric entities to remove from the subset.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add Layer(s) to Subset

You can add layers to a subset using different methods of the **Flood Fill option** as shown below.

Figure 112: Modify Subset – Add Layer Method



- **Add Layer Method**

  Enter the number of layers attached to the selected entity to be added to the subset.

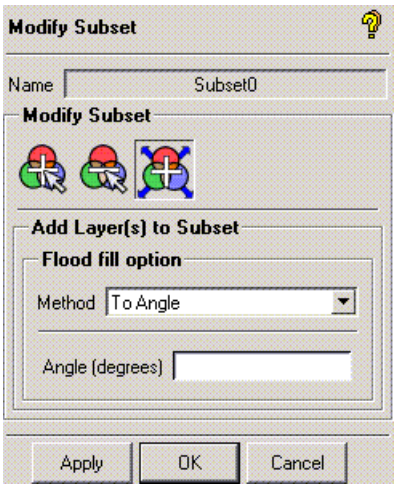
- **All Attached Method**

  This will make a subset of all the attached items of the selected item.

- **To Angle**

  The items that are attached to the selected entity at this angle or less will be added to the subset. Enter the required angle (degrees) as shown below.

  Figure 113: Modify Subset – To Angle Method



- **At one side**

  If an entity is selected, all the surfaces attached to that entity will be highlighted. Select any of the highlighted surfaces with the left mouse button and click the middle mouse button or press Apply to add that surface to the subset.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
Deletes the subset

Rename
To rename the subset.

Copy
Makes a copy of the subset.

Clear
Clears the subset contents.

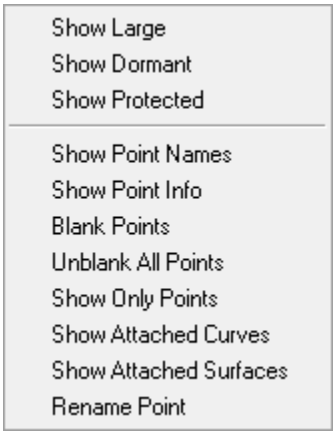
Add
To add elements to the subset.

Remove
To remove elements from the subset.

Geometry Points Options

The display options for Points are shown below.

Figure 114: Points Display Options



Show Large
Shows the visible points as large points.

Show Dormant
Shows all the dormant points which are not permanently deleted.

Show Protected
This option will show a yellow sphere around protected points, a red sphere around hard-protected points.

Show Point Names
Displays the point names of the visible points.

Show Point Info
Gives information about a selected point, including the point name, part name, and geometric location.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To employ this option, select **Points > Show Point Info**. Proceed to select a point with the left mouse button and accept the selection with the middle mouse button. Information on the point(s) will be listed in the **Messages** window.

Blank Points
Blanks selected points.

Unblank All Points
Restores blanked points.

Show Only Points
Displays only the selected points and makes everything else invisible.

Show Attached Curves
Displays the curves which are attached to the selected point.

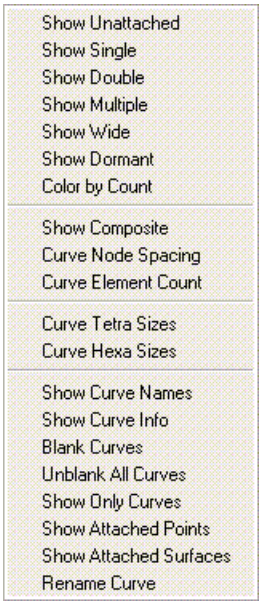
Show Attached Surfaces
Displays the surfaces which are attached to the selected point.

Rename Point
To rename selected points.

Geometry Curves Options

The display options for Curves are shown below.

Figure 115: Curves Display Options



Show Unattached
Shows the curves that are not attached to any of the geometry.

Show Single
Shows all the curves that are attached to only one surface.

Show Double
Shows all the curves that are shared by two surfaces.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Multiple
Shows all the curves that are shared by more than two surfaces.

Show Wide
Displays line data with thicker lines, to help distinguish between geometry and mesh.

Show Dormant
Shows all the dormant curves which are not permanently deleted.

Color by Count
Colors curves by the number of surfaces the curve is associated with. This option works only after CAD repair. See Repair Geometry (p. 241).

- Green

  Not associated with any surfaces

- Yellow

  Associated with only one surface edge

- Red

  Associated with two surface edges

- Blue

  Associated with more than two surface edges

Show Composite
Displays the existing composite curves with different colors. Blocking must be loaded for this option. Composite curves are curves that are grouped together for the purpose of edge association. See Blocking > Associate > Group/Ungroup curves.

Curve Node Spacing
Displays a preview of the node spacing on curves.

Curve Element Count
Displays the number of elements prescribed on each curve.

Curve Tetra Sizes
Displays a tetra icon sized to indicate the tetra sizes that are set (if any) on each visible curve.

Curve Hexa Sizes
Displays a hexa icon sized to indicate the hexa sizes that are set (if any) on each visible curve.

Show Curves Names
Displays the names of the visible curves

Show Curve Info
Gives information about a selected curve, including the curve and part names.

To employ this option, select **Curves > Show Curve Info**. Proceed to select a curve with the left mouse button and accept the selection with the middle mouse button. Information on the curve(s) will be listed in the **Messages** window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blank Curves
Blanks all the selected curves.

Unblank All Curves
Restores blanked curves.

Show Only Curves
Shows only selected curves on the screen and keeps others invisible.

Show Attached Points
Shows all the points which are attached to the selected curve.

Show Attached Surfaces
Shows the surfaces which are attached to the selected curve.

Rename Curve
You can rename the selected curves.

Geometry Surfaces Options

The display options for Surfaces are shown below.

Figure 116: Surfaces Display Options

Show Full
A more detailed representation of the CAD data, showing more isobars than in the Simple representation. For triangulated surface data, a detailed representation will show all the surface triangles.

Show Simple
A wireframe representation of CAD data or a "Hard Feature" representation of triangulated surface data, including surface boundaries as hard features.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Simpler
This wireframe representation requires less memory than the Simple option. This option is beneficial for simplifying more complicated models, and making them easier to translate or rotate.

Show Simplest
This representation uses the least memory. This option is recommended for simplifying models that are extremely complicated.

Wire Frame
The surfaces are drawn using lines, in either simple or detailed formats.

Solid
A solid or shaded representation of the surface data, which can be either simple or detailed.

Solid & Wire
A combination of the **Solid** and **Wireframe** representations of the surface data.

Grey Scale
This will convert the color geometry into grey scale.

Transparent
This will show the surface as transparent.

Show Surface Normals
When enabled, arrows indicating the positive normal direction of each visible surface will be displayed.

Color by Normal
When enabled, the part color is used to indicate the positive normal side of each visible surface. The reverse side of each surface is colored gray.

Note

This option works only with **Solid** or **Solid & Wire** display.

Show Surface Thickness
This will show the thickness assigned for the surface.

Tetra Sizes
Selecting this option will display the reference tetra mesh sizes upon the selected visible surfaces.

Hexa Sizes
Selecting this option will display reference hexa sizes upon the selected visible surfaces.

Show Surface Names
Displays the names of the visible surfaces.

Show Surface Info
Gives information about a selected surface, including the surface and part names.

To employ this option, select **Surfaces>Show Surface Info**. Proceed to select a surface with the left mouse button and accept the selection with the middle mouse button. Information on the surface or surfaces will be listed in the **Messages** window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blank Surfaces
Blanks the selected surfaces.

Unblank All Surfaces
Restores blanked surfaces.

Show Only Surfaces
Displays the selected surfaces only.

Show Attached Points
Displays points that are attached to the selected surface.

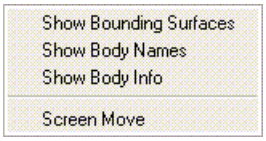
Show Attached Curves
Displays curves that are attached to the selected surface.

Rename Surface
To rename the selected surfaces.

Geometry Bodies Options

The display options for Bodies are shown below.

Figure 117: Bodies Display Options



Show Bounding Surfaces
Shows the boundary surfaces of the selected body.

Show Body Names
Displays the names of the visible bodies.

Show Body Info
Gives information about a selected body, including the body and part names.

To employ this option, select **Bodies > Show Body Info**. Proceed to select a body with the left mouse button and accept the selection with the middle mouse button. Information on the body will be listed in the **Messages** window.

Screen Move
Allows interactive movement of the material point in the plane of the screen.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Densities Options

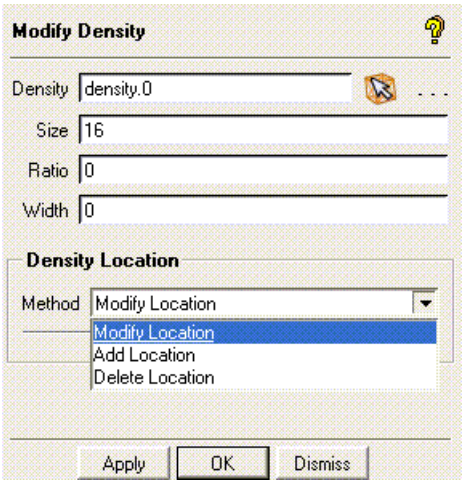
Figure 118: Density Options Window

Create Density

Opens the **Create Density Window** which is explained in Create Mesh Density (p. 327).

Modify Density

Allows you to modify a previously created density region. The following options are available:



Size

Specifies the local maximum mesh size specified for the density region. This will be multiplied by the **Global Scale Factor**.

Ratio

Specifies the tetra growth ratio away from the density region.

Width

For a density region, this specifies the number of layers (N) of the specified element size away from the boundary of the density region that should have a constant expansion ratio. The layer N + 1 will have a tetra size of the **Size** value multiplied by the **Ratio**.

For line and point densities, the **Size** value multiplied by the **Width** is the radius of the region that the density region influences.

Density Location

Contains options for modifying the bounding nodes of the density region.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Location
Allows you to select a new location for an existing bounding node. Select the node to be moved and its new location.

Add Location
Allows you to add new bounding node(s) to the existing density region definition. Select the location(s) to be added to the density region extents and confirm the selection using the middle-mouse button.

Delete Location
Allows you to delete existing bounding node(s) from the density region definition.

Delete Density
Allows you to select the density region(s) to be deleted.

Wide Density Lines
Displays wider density region lines.

Note

This option is often useful when taking images of the mesh setup.

Density Tetra Sizes
Shows the reference tetra size for the density region(s).

Density Color
Allows you to display the density region(s) in the color of your choice.

Show Density Names
Shows the names of the density region(s) on the screen.

Show Density Info
Reports information about the selected density region(s) in the message window.

Blank Densities
Blanks the selected density region(s).

Unblank All Densities
Restores blanked density region(s).

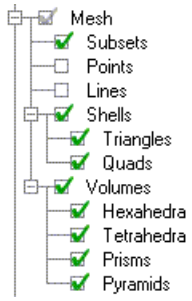
Rename Density
Allows you to rename the selected density region.

Mesh

The **Mesh** branch of the Display tree allows you to display all or part of the mesh by element type.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

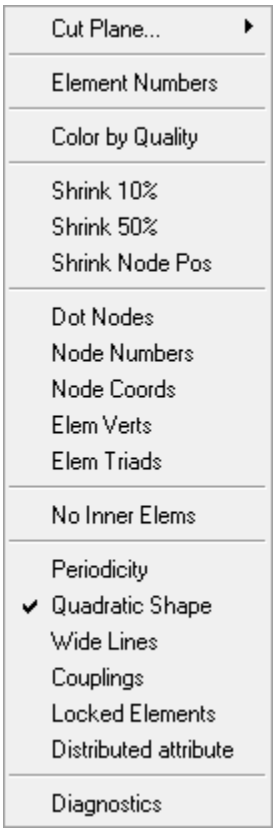
Figure 119: Mesh Tree



Mesh Display Options

Right click on **Mesh** in the Display tree to see the options available for displaying the mesh, shells or volumes, as shown below.

Figure 120: Mesh Display Options



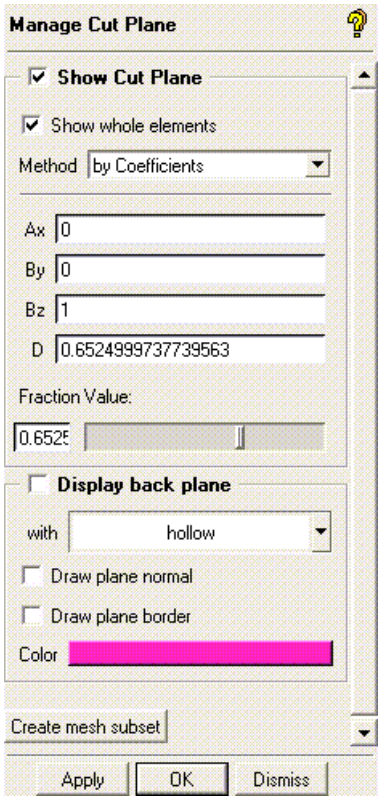
Cut Plane

- Manage Cut Plane

  Opens the **Manage Cut Plane** DEZ as shown below. This option is described in Mesh Cut Plane (p. 62) in the Main Menu chapter.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 121: Manage Cut Plane Window



- Show Cut Plane

  If toggled ON, the Cut Plane will be displayed.

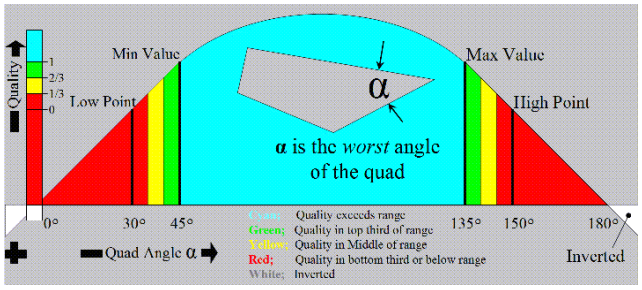
Element Numbers

Displays the numbers of the visible elements.

Color by Quality

Displays the elements according to the quality color setup. The color contour bar displays the range of colors by quality.

Figure 122: Example of Color by Quality for Quality Quad Angle



Shrink 10%

Displays each element shrunk by 10% of its original size.

Shrink 50%

Displays each element shrunk by 50% of its original size.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shrink Node Pos
Works in collaboration with the Shrink features that were previously described to redraw the nodes, depending on whether or not the elements were shrunk.

Dot Nodes
Displays nodes as dots. If the display background color is dark, the dot nodes will be displayed as white, and if the background color is light, the dot nodes will be displayed as black.

Node Numbers
Displays the numbers of the visible nodes.

Node Coords
Displays the coordinates of all the visible nodes.

Elem Verts
Displays the node order numbers of the visible elements. Works best if a shrink factor is also displayed. The element vertex numbers can help you see how an element is defined, and thus see the orientation of the element. For example, a tri element is defined by three nodes in a particular order (0, 1, 2). This defines the orientation of the element.

Elem Triads
Displays the orientation triad of each visible element, with the I, J, and K directions.

Periodicity
Identifies the periodicity of the nodes, either translational or rotational.

Note

Periodicity applies only to Volume (Tetra and Hexa) Meshing, as well as Patch Independent Meshing.

Quadratic Shape
Displays the correct curvature of edges with midside nodes.

Wide Lines
Displays wider lines for elements.

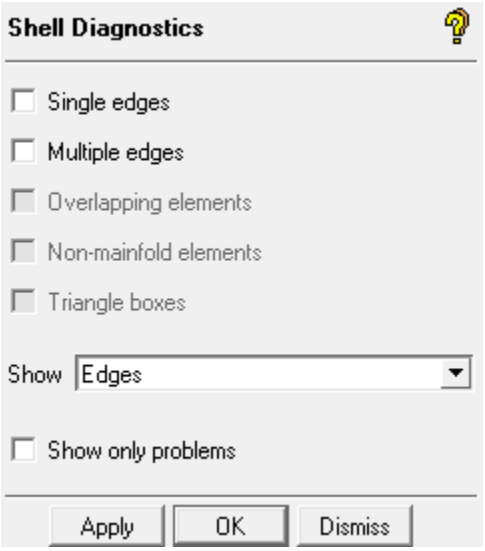
Couplings
Displays couplings. Couplings consists of defined hanging nodes and arbitrary interfaces.

Locked Elements
Displays any locked elements.

Distributed attribute
Makes the element with a distributed attribute visible. For example, if a model has distributed boundary conditions, selecting this option allows you to view and edit the BC at an element or node basis. This option is also available as an icon on the **Edit Mesh** toolbar.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diagnostics



This option allows you to utilize shell mesh diagnostics. The mesh diagnostic options can also be accessed through **Edit Mesh > Check Mesh**. The difference is that the diagnostic options in the Mesh Display Tree does not add problematic mesh to mesh subsets.

Single Edges

Displays any single edges of surface mesh elements that are visible. A single edge would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

Multiple Edges

Displays any edge that is shared among three or more surface elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet. These elements are a subset of the single and multiple edge checks.

Overlapping elements

Displays surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself.

Non-manifold elements

Displays surface elements with non-manifold vertices. Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the mesh that probably should not be there.

Triangle boxes

Groups of four triangles that form a tetrahedron with no volume element inside.

Show

Edges

Displays only the edges of the surface mesh that are found by the selected diagnostic options. Single edges will be displayed in yellow, and multiple edges in blue.

Edges and Faces

Displays the edges and one layer of attached faces.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edges and 2 layers of Faces
Displays the edges and two layers of attached faces.

Show only problems
Displays only the elements found by the diagnostic options and turns off the rest of the mesh.

Mesh Subsets

Elements can be grouped into **Subsets**, which have display options that can be controlled independently.

Note

To view the contents of a single subset, select the subset and deselect all other subsets.

Subsets work independently of the active elements under parts and types. You can switch between displaying entities by part, type, or by a defined grouping in a subset. Parts and Subsets are Boolean with the geometry, but they are not Boolean with each other, meaning, by turning a subset off, it does not turn those entities off unless they are off in the part as well. To access specific entities of geometry or mesh, you can place the entities into a subset. By turn off the parts, and displaying the subset, you can choose to view only the entities in the subset.

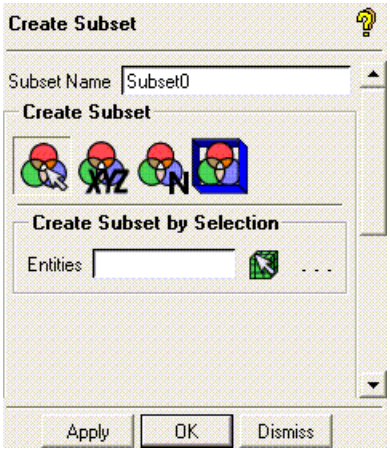
Parts are exclusively defined, so that an entity can only be in one part. But one entity can be part of multiple subsets.

Clicking the right mouse button on **Subsets** in the Mesh Tree will give the options below.

Figure 123: Mesh Subsets Display Options

Create
To create a new subset of currently active parts and types. There are different options for creating subsets as shown below.

Figure 124: Create Subset Window



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

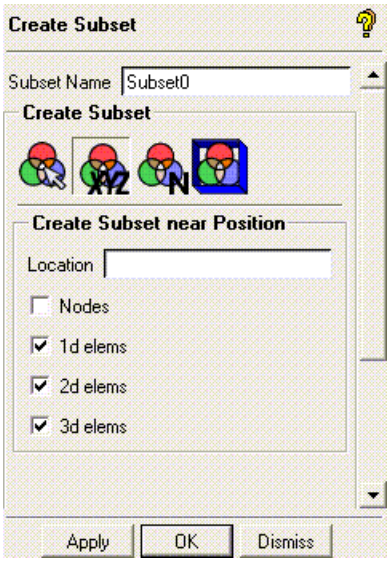
Create Subset by Selection

Create a subset by entering a name and selecting either geometric or mesh entities.

Create Subset near Position

Create a subset near a certain position by entering the XYZ coordinates as shown below.

Figure 125: Create Subset Near Position Window



- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Create Subset by Element Number

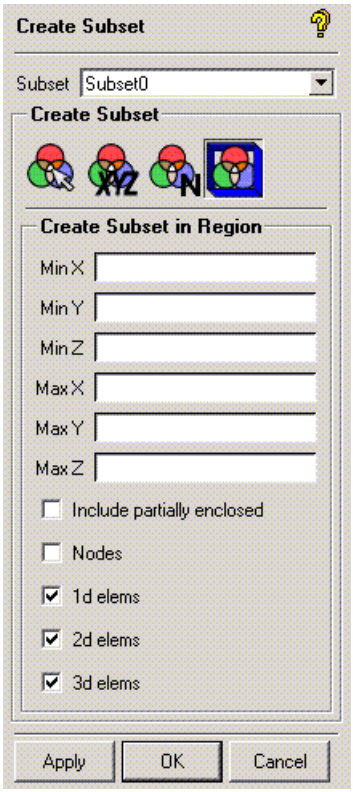
Create a subset by entering the elements numbers of the elements you want to include.

Create Subset in Region

Create a subset in a specified region as shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 126: Create Subset in Region Window



- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represents the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Find Worst

Creates a subset of the elements with the worst quality found using the "Quality" criterion.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show All
Displays all subsets.

Hide All
Hides all subsets.

Shadings
allows you to set the shading for the subset independent of the shading used for the mesh. This allows for better visibility as the subsets can be distinguished from the surrounding mesh. By default, the shading of the subset is **Inherited** from the shading used for the mesh. You can also select the **Wire Frame**, **Solid**, or **Solid & Wire** rendering.

Note

Subsets created from the Quality Metric Histogram will retain their **Color by Quality** based color and shading independent of this setting.

Tip

- To display other subsets with **Color by Quality**, combine the **Solid** or **Solid & Wire** subset shading with the **Color by Quality** mesh display option.
- To display the elements of a **DIAGNOSTIC** subset without the **Color by Quality** shading, you can copy it to a new subset and disable the **DIAGNOSTIC** subset (**Mesh > Subsets > DIAGNOSTIC**).

Modify Mesh Subsets

Right clicking on an existing mesh subset displays the options shown below.

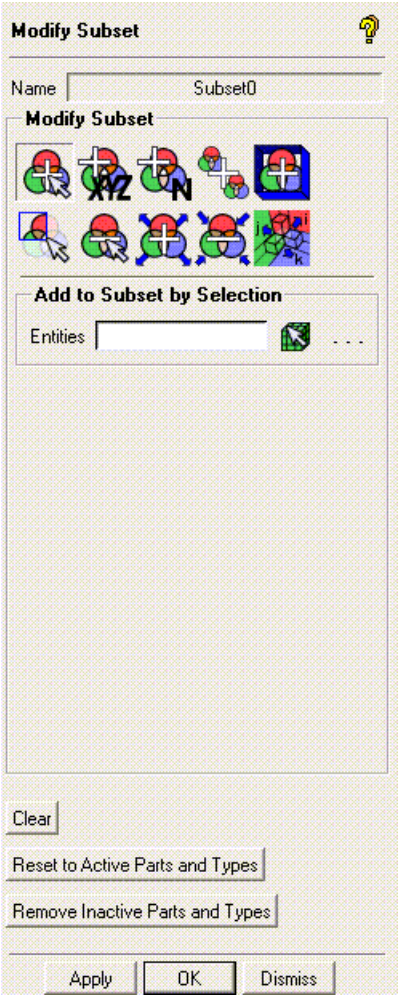
Figure 127: Modify Mesh Subset Options

Info
Gives information about the subset.

Modify
Subsets can be modified by the options in the window shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 128: Modify Subset Window



The following options are common to all of the **Modify Subset** options.

- **Clear**

  Clears all the entities in the subset.

- **Reset to Active Parts and Types**

  Resets the subset to contain all active (visible) parts and types.

- **Remove Inactive Parts and Types**

  Removes the inactive (not visible) parts and types from the subset.

The different **Modify Subset** options are described below.

Add to Subset by Selection

Select mesh entities to add to a subset.

Add to Subset near Position

Enter the node numbers or XYZ coordinates of entities to add to a subset.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add by Element Number

Enter element numbers to add to a subset.

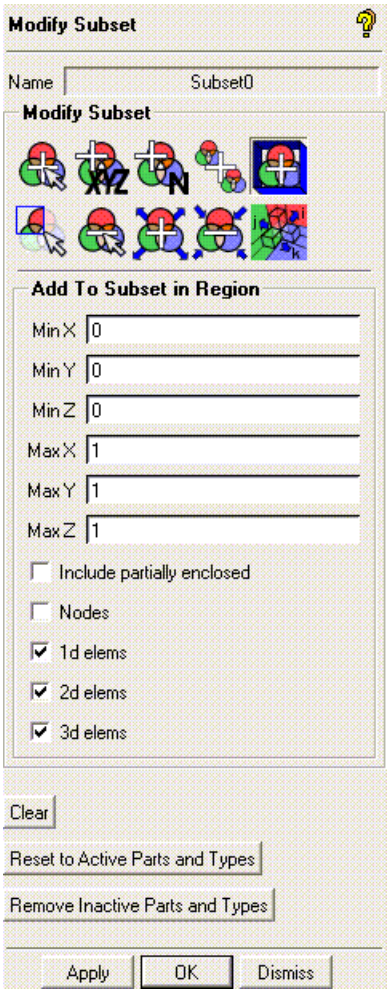
Add Contents of Subset

Add contents of an existing subset to the current subset.

Add to Subset in Region

Choose a subset with a specified region to modify.

Figure 129: Add to Subset in Region Window



- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represent the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

Restrict to subset by selection

Allows you to restrict a subset to the entities that are selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

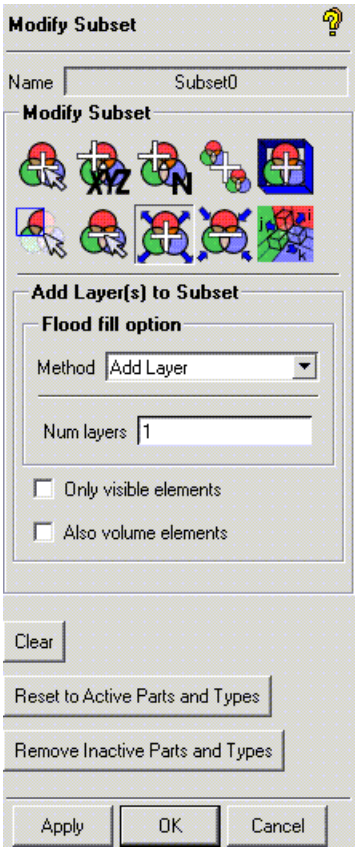
Remove from subset by selection

Removes the selected entities from the subset.

Add Layer(s) to Subset

You can add layers to a subset using different methods of the **Flood Fill option** as described below.

Figure 130: Add Layer(s) to Subset Window



- **Add Layer Method**

  Enter the number of layers attached to the selected entity to be added to the subset.

- **All Attached Method**

  This will add all the attached items of the selected entity to the subset.

- **Same Part**

  Adds all entities in the same part.

- **To Curve**

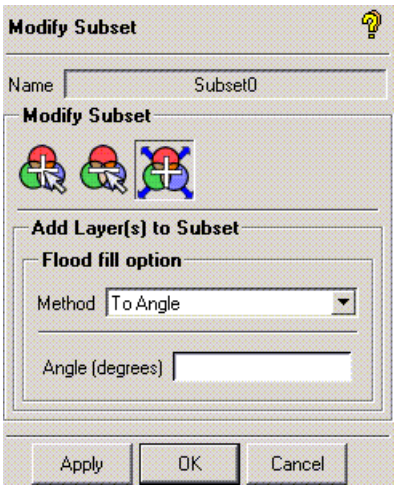
  Adds all entities up to the nearest boundary curve.

- **To Angle**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The items that are attached to the selected entity at this angle or less will be added to the subset. Enter the required angle (degrees).

Figure 131: Modify Subset – By Angle Method



– **Only visible elements**

Toggle **ON** to add visible entities only.

– **Also volume elements**

Toggle **ON** to add volume elements.

Remove Layer(s) from Subset

You can remove layers from a subset using different methods as described below.

- **Num Layers**

  Enter the number of layers to be removed.

- **Only visible**

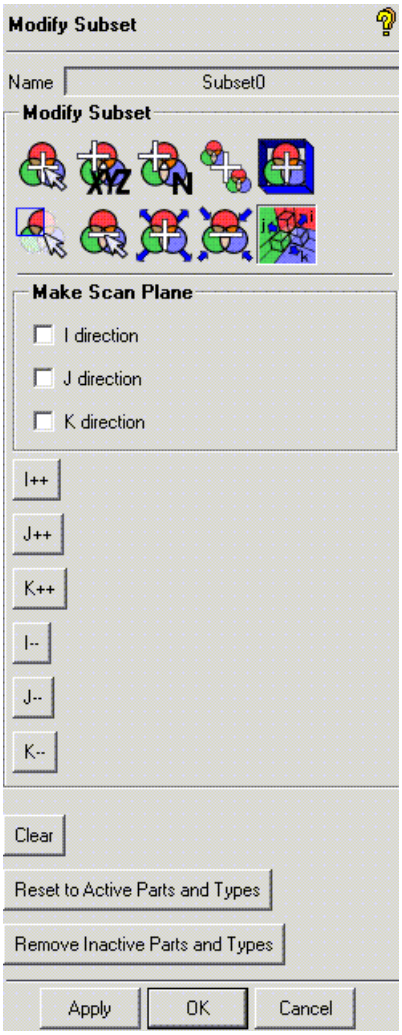
  To remove visible entities only.

Make scan plane

Creates a subset of scan plane elements. This option is for advanced users.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 132: Make Scan Plane Window



Delete
Deletes an existing subset.

Rename
To rename a subset.

Copy
Copies a subset.

Clear
Clears the contents of a subset.

Add 1 layer
Adds the adjacent layer of entities to the subset.

Add
To select entities to be added to the subset.

Remove
To select entities to be removed from the subset.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make Element set
To define a subset as an element set.

Make Node set
To define a subset as a node set.

Mesh Points

Click on **Points** in the Mesh Tree to turn nodes **ON** or **OFF** from the display.

Mesh Lines

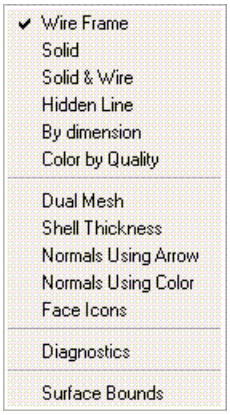
Click on **Lines** in the Mesh Tree to turn bar elements **ON** or **OFF** from the display.

Right clicking on **Lines** in the Mesh Tree gives the display option **Direction**. Toggling ON this option displays an arrow showing the relationship between the first and second nodes along the line element. This can be useful to display the dependency for 1D line element properties. The direction of the line element can be changed using **Edit Mesh > Reorient Mesh > Reverse Line Element Direction**.

Mesh Shells

The display options for **Shells** are shown below.

Figure 133: Shells Display Options



Wire Frame
Displays the mesh with a wire frame outline, colored part by part.

Solid
Displays the mesh as a solid mesh.

Solid & Wire
Displays the mesh as a solid mesh with a wire frame outline. The wire frame is drawn in the color of the background.

By dimension
Displays shell elements only as solid mesh.

Hidden Line
Displays a wire frame mesh with the backside of the model hidden.

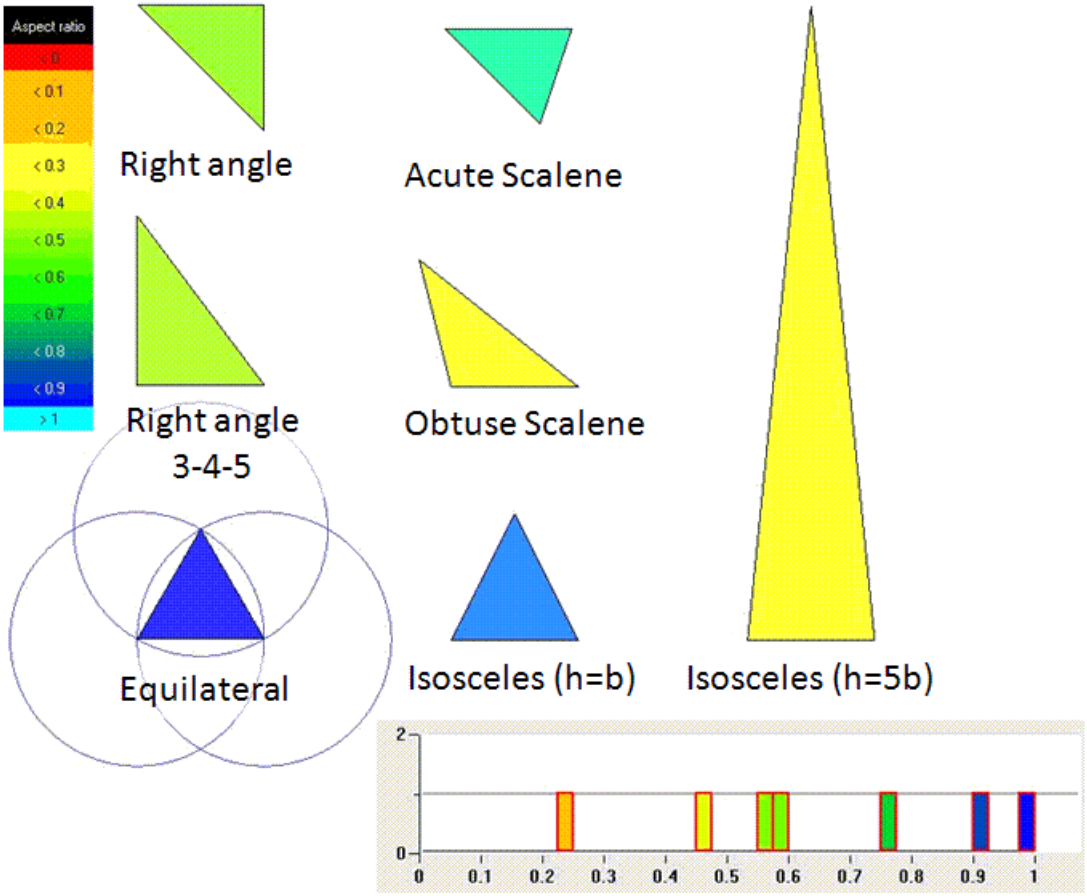
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color by Quality

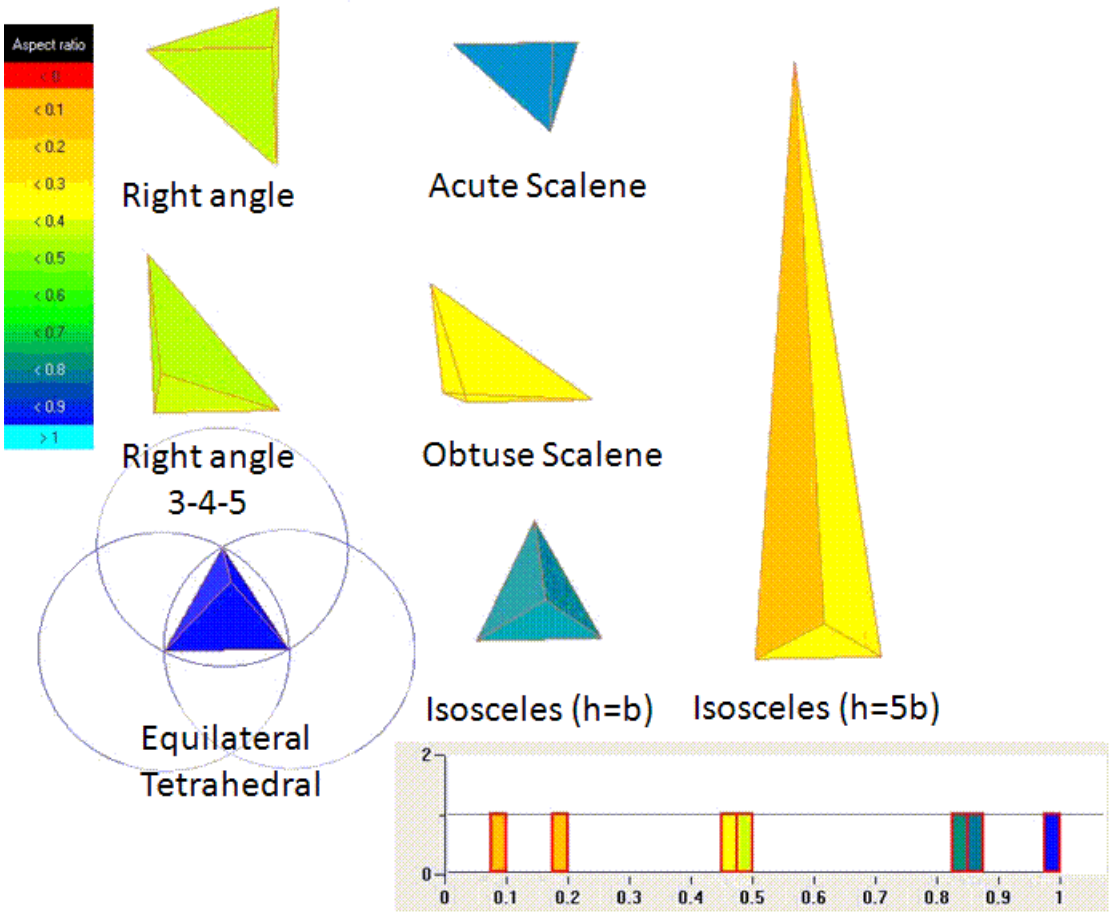
Displays the shell elements according to the quality color setup. The color contour bar displays the range of colors by quality.

Figure 134: Examples of Color by Quality—Aspect Ratio

Tri elements



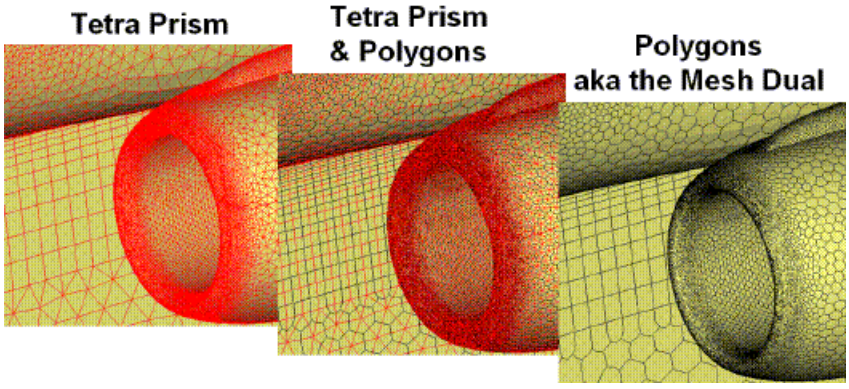
Tetra elements



Dual Mesh

Allows you to view the mesh dual. When enabled, it draws the wire-frame between the centroid of each cell to better show the node centered volumes. An example of the mesh dual for a tetra-prism mesh is shown in Figure 135: Mesh Dual for a Tetra-Prism Mesh (p. 156).

Figure 135: Mesh Dual for a Tetra-Prism Mesh



This option is just a visual aid and can be used only for surface meshes.

Note

The mesh dual of a hexa mesh is still a hexa mesh.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shell Thickness
Displays the surface mesh thickness.

Normals Using Arrow
When enabled, arrows indicating the positive normal direction of each visible unstructured surface element will be displayed.

Normals Using Color
When enabled, the part color is used to indicate the positive normal side of each visible unstructured surface element. The reverse side of each element is a darker color.

Note

This option works only with **Solid** or **Solid & Wire** display.

Face Icons
Adds part colored face icons to the wireframe display. The icons appear similar to **Shrink 75%**.

Diagnostics
This option allows you to utilize shell mesh diagnostics. The mesh diagnostic options can also be accessed through **Edit Mesh > Check Mesh**. The difference is that the diagnostic options in the Mesh Display Tree does not add problematic mesh to mesh subsets.

Single Edges
Displays any single edges of surface mesh elements that are visible. A single edge would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

Multiple Edges
Displays any edge that is shared among three or more surface elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet. These elements are a subset of the single and multiple edge checks.

Overlapping elements
Displays surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself.

Non-manifold elements
Displays surface elements with non-manifold vertices. Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the mesh that probably should not be there.

Triangle boxes
Groups of four triangles that form a tetrahedron with no volume element inside.

Show

Edges
Displays only the edges of the surface mesh that are found by the selected diagnostic options. Single edges will be displayed in yellow, and multiple edges in blue.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edges and Faces
Displays the edges and one layer of attached faces.

Edges and 2 layers of Faces
Displays the edges and two layers of attached faces.

Show only problems
Displays only the elements found by the diagnostic options and turns off the rest of the mesh.

Surface Bounds
Displays the surface boundary region and all elements attached to that region. Useful in visualizing curve boundary regions.

Mesh Volumes

Volume display options are shown below.

Figure 136: Volume display options

Wire Frame
Displays the elements with a wire frame mesh on it.

Solid
Displays the elements as a solid mesh.

Solid & Wire
Displays the mesh as a solid mesh with a wire frame outline. The wire frame is drawn in the color of the background.

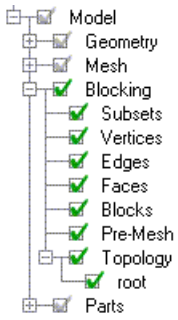
Hidden Line
Displays a wire frame mesh with the backside of the model hidden.

Volume Bounds
Displays the volumetric boundary region and all elements attached to that region. Useful in visualizing the surface boundary regions.

Blocking

Click the right mouse button on **Blocking** in the Display tree to see the options available.

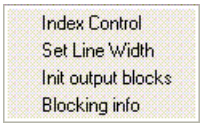
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Blocking Options

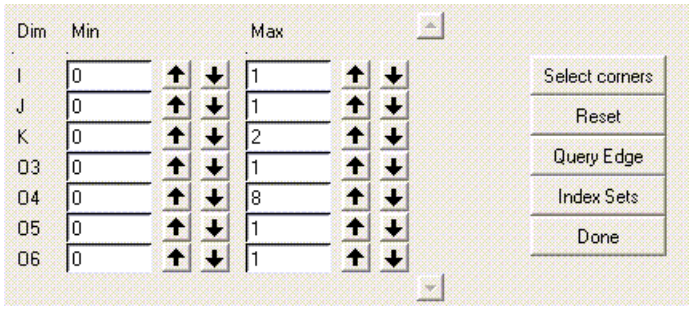
Right-click on the Blocking branch in the Display tree to view the Blocking display options. When the blocking tab is active, the hot key "i" will initialize the index control. The Blocking display options are as follows:

Figure 137: Blocking Display Options



Index Control

Figure 138: Index Control Window



Select corners

Allows you to select boundary node vertices and adjusts the index range accordingly.

Reset

Resets the values to the minimum and maximum values, so that the entire blocking is displayed.

Query Edge

Allows you to select an edge. The system will then center the Index control on the dimension corresponding to the edge selected

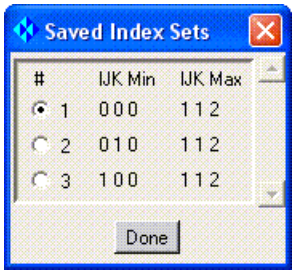
When an O-grid is created, a new index direction will be added to the Index control and a message will be printed with the corresponding dimension. O-grid dimensions are named O_n where n begins with a value of 3. The reason for this is that dimensions 0, 1, and 2 already correspond to the I, J, and K indices.

The index range is not taken into account when the mesh is saved.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index Sets
Contains options for saving and managing index sets based on the index control values.

Show
shows the list of the currently saved index sets in the **Saved Index Sets** dialog.



You can select the index set to restore from the list and the blocking display will be reset corresponding to the saved index values.

Add
saves the current set of index values. The saved index sets are numbered sequentially (**1**, **2**, etc.).

Clear
clears all the currently saved index sets from the **Saved Index Sets** list.

Load
allows you to load previously saved index sets from an index control (*.ictrl) file.

Save
allows you to write the currently saved index sets to an index control (*.ictrl) file.

Done
Closes the window

Set Line Width
Sets the width of Block edge lines.

Init output blocks
Initializes the number of output blocks to be equal to the number of regular topology blocks.

Blocking info
Provides information in the message window about the blocking (number/type of blocks, topology info, index info) and pre-mesh (number of nodes, elements). Information about shared walls (if defined) will also be reported.

Blocking Subset Options

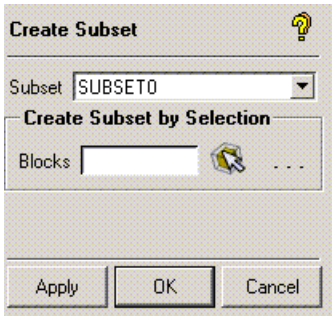
Right click on **Subsets** in the **Blocking** tree to view the Display options are shown below.

Figure 139: Blocking Subset Display Options

Create
Select entities to add to a new Blocking Subset.

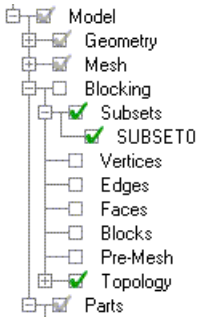
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 140: Create Subset Window



After the subset is created, it will be displayed in the Display Tree under **Blocking>Subsets** as shown below.

Figure 141: Blocking Subset Tree



Modify Subsets Options

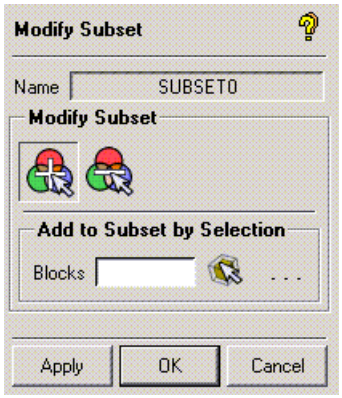
Right click on a created subset to view the options shown below.

Figure 142: Modify Subset Options

Modify

Blocking subsets can be modified with the options shown below.

Figure 143: Modify Subset window



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add to Subset by Selection

Select the blocking entities to add to the subset.

Remove from Subset by Selection

Select the blocking entities to remove from the subset.

Delete
Deletes the subset.

Rename
Renames the subset.

Vertices

The display options for **Vertices** are shown below.

Figure 144: Vertices Display Options

On surfaces
To turn the display of vertices on surfaces **ON** or **OFF**.

Periodic
To turn the display of vertices that are marked as periodic **ON** or **OFF**.

Names
Displays the names of the vertices.

Proj type
Vertices will be tagged according to type:

- **p**: Associated to a prescribed point (red).
- **c**: Associated to a curve (green).
- **s**: Associated to a surface (white).
- **v**: Associated to an interior, volume, or vertex (blue).

Indices
Displays the block indices.

Numbers
Vertices will be displayed with unique numbers assigned to them during creation of the blocks.

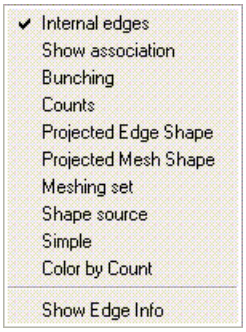
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Vertex Info
Gives information about a selected vertex.

Edges

The display options for **Edges** are shown below.

Figure 145: Edges Display Options



Internal edges
To turn the display of internal edges in a volume **ON** or **OFF**.

Show associations
Shows the associations of different edges to curves.

Bunching
Displays the node distribution on edges.

Counts
Displays the number of elements of the edges.

Projected Edge Shape
Displays the edges after mesh calculation with the current node distribution.

Projected Mesh Shape
Displays the mesh after calculation with the current edge distribution.

Meshing set
Displays the special meshing sets.

Shape source
Turns the linked edges **ON** or **OFF**. All the arrows will point from the source edge to the target edge.

Simple
Shows edges in simple form.

Color by Count
Only for 2D blocking.

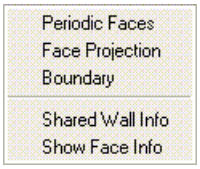
Show Edge Info
Gives information about the selected edge. The edge vertices, spacing, ratio, constraints, edge dimension, index, number of nodes, mesh law, and number of edge segments (if any) will be reported in the message window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faces

The display options for **Faces** are shown below.

Figure 146: Faces Display Options



Periodic Faces

Allows you to turn the display of periodic faces ON or OFF. Periodic faces are established when all associated vertices have been set to periodic using the **Periodic Vertices** option in the **Blocking** tab. Vertex periodicity can also be displayed by enabling **Periodic** for **Vertices** in the tree. Further splits across periodic faces create new edges and vertices which are automatically set to periodic.

Face Projection

Toggles ON or OFF the display of faces associated with surfaces (see Associate Face to Surface (p. 398)).

The projected faces will be displayed depending on their association as follows:

Face Association	Display
Face→Part	Part Color along with Part Name
Face→Link Shapes	White or Black (depending on the background color)
Face→Selected Surface	Purple
Face→Interpolate	Green
Face→Reference Mesh	Yellow

Boundary

Displays only the faces that lie on the outer surfaces.

Shared Wall Info

Reports information about the currently defined shared wall configurations in the message window. The volumes having a shared wall between them and the shared surface will be reported.

Show Face Info

Gives information about the faces. The face vertices, face elements, and the projection type will be reported in the message window.

Blocks

The display options for **Blocks** are shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 147: Blocks Display Options

Solid
Displays blocks in solid color.

Whole blocks
Shows each block as a black icon.

No shrink
If toggled **ON** , the actual size of the block is displayed.

Find Worst
This option calculates the determinant value for each block and highlights the elements with the worst value or values. A bad determinant block usually results in a bad determinant grid. Moving the vertices can help improve the determinant values. You can set the number of worst blocks to display by clicking on **Settings** and choosing **Meshing Options>Hexa Meshing**, then setting a numerical value for **Find Worst**. The default range is 1–3, but you can set the range to any desired level (for example, 3–5 or 2–8). The worst blocks are listed by their determinant values and shown in red in the model.

IJK
Displays the IJK grid orientation for each block.

Refinement
This option will blank all the black icons and will display the blocks that are affected by the **Refinement** command, which is in the Meshing Menu.

Classes
Displays the blocks along with their respective axis and index numbers. For each block the three axial directions are indicated by 1, 2, and 3, and O-grids are represented by 0.

Show structured
Shows the structured blocks.

Show unstructured
Show the unstructured blocks.

Show swept
Shows swept blocks in the geometry.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blanking
Restores blanked blocks.

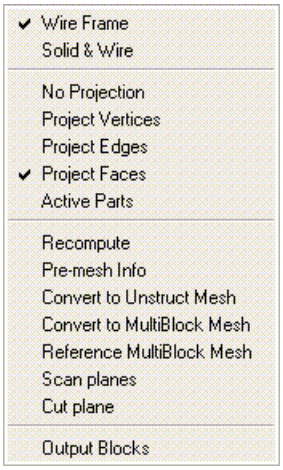
Blank Blocks
Blanks the selected blocks.

Show Block Info
Gives information about the selected block.

Pre-Mesh

The display options for **Pre-Mesh** are shown below.

Figure 148: Pre-Mesh Display Options



Wireframe
Displays the model in the wireframe mode.

Solid
Displays the model in the solid mode.

No projection
When the mesh is generated, it will conform to the blocking without being projected to the actual CAD geometry.

Project vertices
When the mesh is generated, the vertices/points on the edges of the blocks will be projected to the curves or surfaces to which they are associated. No other projection will take place.

Project edges
When the mesh is generated, the nodes on the edges of the blocks will be projected to the curves or surfaces to which they are associated. The face nodes will not be projected to the surfaces, but rather interpolated between the edges. This option is generally used for two-dimensional models. **Project edges** may be used for three-dimensional models as well, as a method to quickly confirm a correct mesh before employing the **Project faces** operation.

Project faces
When the mesh is generated, all face nodes and edge nodes will be projected to their associated curves and surfaces. This option is generally used for three-dimensional models.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active Parts

Allows you to select surface parts to be considered for projection when the mesh is generated. All surface parts highlighted in the selection list will be projected to during the mesh generation.

When you compute the pre-mesh, a message will appear indicating the part(s) to which projection has been disabled.

Recompute

Recomputes the mesh.

Show size info

Reports the number of nodes and elements in the message window.

Convert to Unstruct Mesh

Converts the blocking to an unstructured mesh.

Convert to MultiBlock Mesh

Writes out the multiblock mesh for the current blocking.

Reference MultiBlock Mesh

Automatically references the multiblock mesh in the current directory for meshing the current blocking.

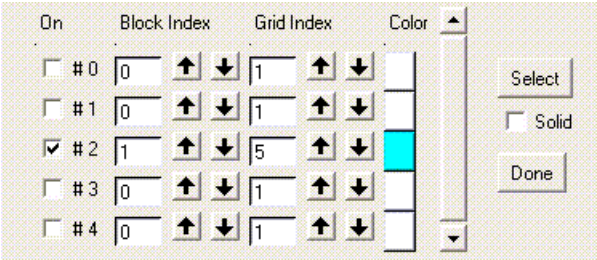
Note

Only one multiblock mesh file should exist in the current directory.

Scan planes

Scan planes are used to view the volume grid. The following Scan plane control window will be displayed.

Figure 149: Scan Planes Window

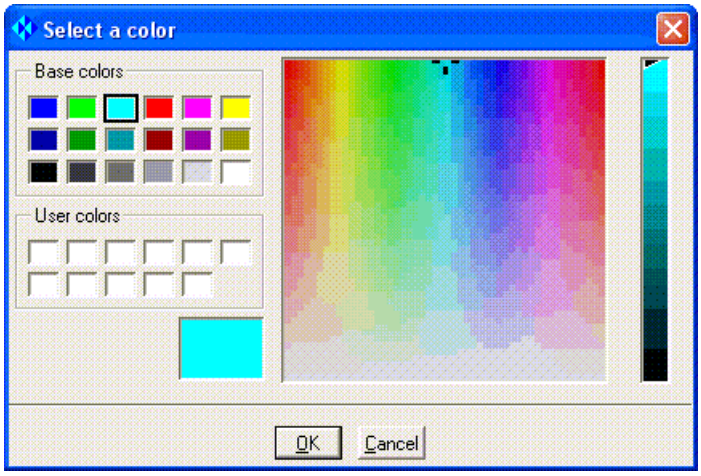


- There are a number of selection fields for displaying scan planes across each of the principal directions (i, j, k, o3, o4, etc.). #0 corresponds to the first scan plane across i, #1 corresponds to the second, across j, etc. Higher scan planes, such as #3 or #4, scan across the first and second O-Grid indices, respectively.
- There is a Block and a Grid index for each scan plane. The Block index is the block number in which the scan plane is displayed. The lowest and highest block are typically in the VORFN region and may not be displayed. The Grid index is the node number within the block.
- The up/down arrow buttons allow you to increase/decrease the Block or Grid index by one, respectively. If you click and hold down the up/down arrow button, you will be able to scroll through the Block/Grid index (you need not click the button repeatedly).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, the scan plane is in the color of the volume parts it passes through. However, the Color buttons allow you to specify a color for each scan plane (see Figure 150: Selecting the Scan Plane Color (p. 168)).

Figure 150: Selecting the Scan Plane Color



- The **Select** button allows you to select an edge in the model. The scan plane perpendicular to this edge is then displayed. This is the easiest way to place a scan plane in the model.
- The **Solid** check box controls whether the mesh will be displayed in solid or wire-frame form.

Tip

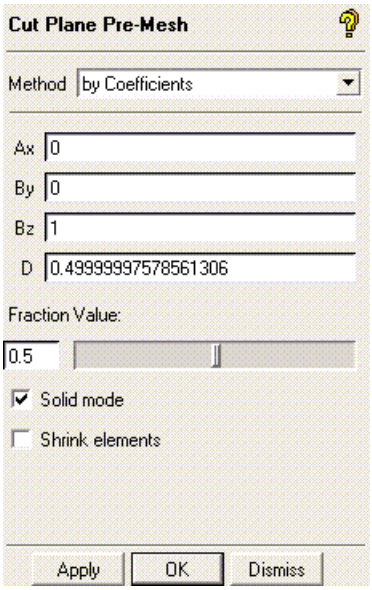
Scan planes are very useful for diagnosing blocking issues. Running a scan plane through the problem area is often the best way to gain understanding of what is happening in the volume. Typically the scan plane will reveal issues with projection or edge distribution. Fixing these will improve the quality. Enabling the display of edge association and edge bunching can also help identify the problem.

Cut Planes

A cut plane is used to visualize results on a plane cut through the three dimensional model. Results are viewed on the cut plane as well as on the 2D Dynamic window. The cut plane may be defined in several ways depending on the application.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 151: Pre-Mesh Cut Plane



Method

- By Coefficients

  Defines the cut plane by the equation of the normal vector. The equation is of the form: Ax + By + Cz + D = 0. Enter the coefficients A, B, C, D of the equation.

- By Point and Normal

  pt

  Enter the Global Cartesian coordinates in the X, Y, and Z direction.

  NX

  Enter the X component of a unit vector normal to the desired cut plane.

  NY

  Enter the Y component of a unit vector normal to the desired cut plane.

  NZ

  Enter the Z component of a unit vector normal to the desired cut plane.

- By Corner Points

  Creates a cut plane passing through the given three points. You can enter the Cartesian coordinates of Pt1, Pt2, and Pt3 points.

- By 3 Points

  You can select any three points in the GUI using the left mouse button to define a cut plane.

- Move or Rotate

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can interactively move a previously defined cut plane while holding the left mouse button down. Once you get to the desired location, release the mouse button. Use the middle mouse button to rotate the cut plane about the normal axis. When you get the desired orientation, release the mouse button. To end the interactive movement of the cut plane, click on the right mouse.

- Middle X plane

  This will put the cut plane at the middle of the geometry in the X-direction.

- Middle Y plane

  This will put the cut plane at the middle of the geometry in the Y-direction.

- Middle Z plane

  This will put the cut plane at the middle of the geometry in the Z-direction.

Fraction value

Select the location of the plane by changing the fraction value either by moving the sliding bar or entering a value.

Solid Mode

To display the cut plane in solid mode.

Shrink elements

To shrink elements in the display.

Note

Cut planes show the grid lines relative to planes in 3D space. Scan planes show grid lines relative to grid planes. Either method may be used. For unstructured blocking, it is recommended to use cut planes.

Note

The cut planes for Pre-Mesh can be viewed before writing out the mesh. This allows for mesh quality inspection and block editing before writing out the final mesh.

Output Blocks

Controls the number of block files written out for multi-block structured grids.

Hexa supports two levels of block decomposition. The first level is what you see by default, and what is used to mesh the model. However, it is sometimes desirable to mesh with one set of blocks and group blocks together for a smaller number of blocks for the solver. The output blocks option can be used to achieve this. First, click on **Init output blocks**. This makes your output blocks the same as your regular blocks.

Then select **Output Blocks** under the Blocking > Pre-Mesh Display Tree. Now, operations such as Merge Blocks, Extend Split, Merge Face, and Renumber Blocks can be performed on the output blocks. By merging blocks you can reduce the number of blocks that are written to the solver. The

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

blocks that are used to mesh the model are unchanged. You can switch between the regular blocks and the output blocks at any time.

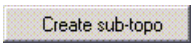
Note

Face selection operations for Output Blocks are not fully supported.

Topology

The display option for **Topology** is shown below.

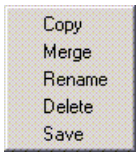
Figure 152: Topology Display Options



Create sub-topo

A portion of the current (root) block topology may be extracted to create a new topology. Each sub-topology has the following display options.

Figure 153: Sub-topo Display Options



- **Copy**

  Makes a copy of a sub-topology.

- **Merge**

  Merges selected sub-topologies.

- **Rename**

  To rename the sub-topology.

- **Delete**

  Deletes the sub-topology.

- **Save**

  Saves the sub-topology.

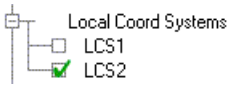
Local Coordinate Systems

You can define local coordinate systems (LCS) for use in geometry, mesh, blocking or boundary condition manipulation using the LCS icon in the main menu. The default coordinate system located at the origin is called Global.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a LCS is defined, its name will appear in the display tree as shown below. Deactivating all Local Coordinate Systems in the Display tree will activate the Global coordinate system. You can view only one LCS at a time.

Figure 154: Local Coordinate Systems Tree



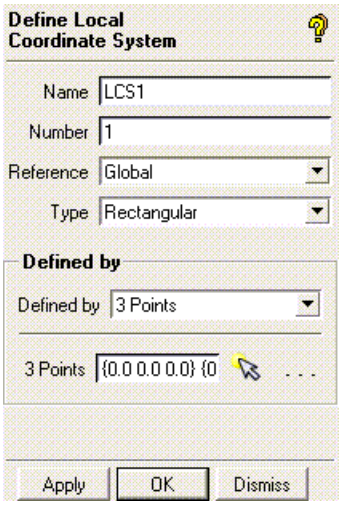
Right click on a defined coordinate system in the Display Tree for the following options.

Delete

Removes the LCS from the display tree and its data from the geometry and parameter files.

Modify

Opens the Define Local Coordinate System DEZ to allow you to modify the coordinate system.



See Local Coordinate Systems in the Main Menu chapter for the description of all the options.

Rename

Opens a dialog box where you may give the LCS a new name.

Flip Axis

Cycles the LCS axes through the sequence X-Y-Z, to Z-X-Y, to Y-Z-X, back to X-Y-Z relative to the original LCS. For non-rectangular coordinate systems the transformation is the same, except using the R, θ, and Z axes, or the R, θ, and Φ axes.

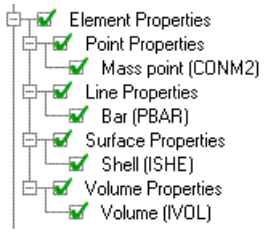
This approach is useful to transform a Z-dominant coordinate system to an X-dominant or Y-dominant coordinate system. For example, if you want to change the orientation for the 1-point definition method or have the first two points of the 3-point method define something other than Z-direction, it might be best to define the LCS in the easiest method possible and Flip Axis to get the appropriate LCS.

Element Properties

If the model contains different defined Elements, such as 0D, 1D, 2D (TRI/QUAD) or 3D elements (TETRA/HEXA), they will be under Points, Lines, Surfaces and Volumes respectively. The Element Properties Tree will appear as shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 155: Element Properties Tree



Points contain all the 0D element properties and Lines contains 1D element properties, etc. When element properties are defined, they will appear in both the Element Properties tree as well as the part display tree to which they belong. To edit any of these definitions, double click the defined properties.

Right clicking on **Element Properties** in the Display tree will give you the options to **Delete Empty Properties**. This option removes empty element properties from the display tree and refreshes the Element Properties branch. This option can be run whenever needed after modifications are made to Element Properties

Connectors

If connectors are defined for a model, they will be listed under the Connectors Tree.

Right clicking on the Connectors Tree will give you the following options.

Figure 156: Modify Connectors Options

Create
Opens the Define Connectors window. See **Mesh > Define Connectors** for more information.

Make All Active
Makes all connectors active.

Make All Inactive
Makes all connectors inactive.

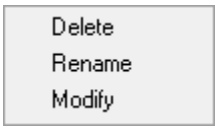
Delete All
Deletes all connectors defined in the model.

Constrained Nodes

Node constraints may be set using the **Define Constrained Node sets** option on the **Constraints** tab, or by right-clicking **Constrained nodes** in the display tree. See the Define Constrained Node Sets page for information on the node constraint options.

If constrained node sets are defined, they will be listed under Constrained nodes in the display tree. Right-clicking on the name of a Constrained Node set, will give you the following options

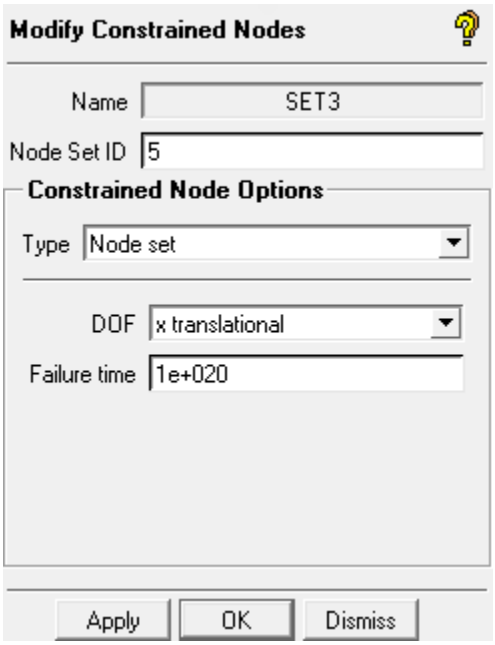
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Delete
Removes the selected constrained node set.

Rename
Opens a dialog box to specify a new name for the constrained node set.

Modify
Opens the Modify Constrained Nodes DEZ where you can change the parameters of the node set. For more information regarding the options available, see the Define Constrained Node Sets page.



Contacts

Contact boundary conditions may be set using the **Define Contact** option on the **Constraints** tab, or by right-clicking **Contacts** in the display tree. See the Define Contact page for information on the contact constraint options.

When you turn on the display for a contact set, the contact area will be drawn as white and the target set as red. This display is useful for Automatic Contact Setup to see the elements that are marked depending on the defined contact proximity factor.

Figure 157: Contacts Tree

To edit a contact boundary condition, right click on the defined contact boundary condition set to choose from the following options.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 158: Modify Contacts Options

Modify
Opens the **Modify Surface-to-Surface Contact Parameters** DEZ. For more information on the options available, see the Define Contact page.

Switch contact/target
To switch the target set and the contact set for the given contact region.

Delete
Deletes the selected contact.

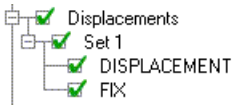
Rename
To rename the selected contact.

Displacements

Multipoint Constraint Equations may be set using the **Create Constraint Equation** option on the **Constraints** tab. See the Create Constraint Equation page for information on multipoint constraint options.

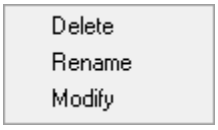
Multipoint Constraint Equations that are defined on points, curves and surfaces will appear in the Display tree as shown below.

Figure 159: Displacements Tree



To edit any constraints, right-click on the constraint name in the display tree and choose from the following options.

Figure 160: Modify Displacement Options



Delete
Removes the selected displacement.

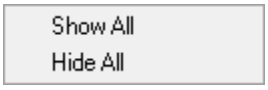
Rename
Opens a dialog box to specify a new name for the displacement.

Modify
Opens the Modify Constraint Equation DEZ where you can change the parameters of the multipoint constraint equation (displacement). For more information regarding the options available, see the Create Constraint Equation page.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right click on **Displacements** in the Display Tree to choose from the following options.

Figure 161: Displacement Display Options



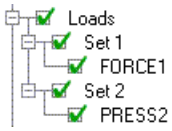
Show All
Displays all the defined displacement.

Hide All
Hides all the displacement.

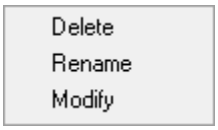
Loads

Loads that are defined on points, curves and surfaces will appear in the Display tree as shown below. In this context, 'Loads' may be of force or pressure type.

Figure 162: Loads Display Tree



Right click on a defined load to choose from the following options.



Delete
Deletes the selected load.

Rename
Opens a dialog box to specify a new name for the selected load.

Modify
Opens a **Modify Force** (or **Modify Pressure**, as appropriate) DEZ which allows you to adjust the parameters of the load.

For full information on the options available, see Create Force (or Place Pressure) in the Help viewer.

Right click on **Loads** in the Display Tree to choose from the following options.

Show All
Displays all the defined loads.

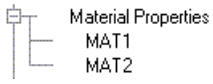
Hide All
Hides all the loads.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

Material Properties that are defined are located in the Display tree as shown below.

Figure 163: Material Properties Display Tree



To edit the material properties of an element, you can double click the left mouse button on the defined material name in the Display Tree to open the **Define Material Property** DEZ.

Right clicking on a defined material name in the Display Tree will give you the following options.

Figure 164: Modify Material Properties Options

Delete
Removes the Material Property definition.

Rename
Opens a dialog box to assign a new name to the selected material property.

Modify
Opens the **Define Material Property** DEZ to allow to modify the material properties.

For more information on the options available, see Create Material Property in the Help viewer.

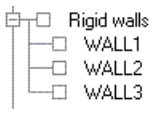
Copy
Opens a dialog box to assign a distinct name to the new material being defined. The new material properties may then be modified as necessary.

Rigid Walls

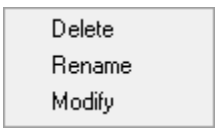
Rigid Wall boundary conditions may be set using the **Define Planar Rigid Wall** option on the **Constraints** tab, or by right-clicking **Rigid walls** in the Display tree. See the Define Planar Rigid Wall page for information on the Rigid Wall boundary condition options.

When set, Rigid Wall boundary conditions will appear in the Display Tree as shown below.

Figure 165: Rigid Walls Tree



Right-clicking on the name of a Rigid wall, will give you the following options.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete
Removes the selected rigid wall boundary condition.

Rename
Opens a dialog box to specify a new name for the rigid wall boundary condition.

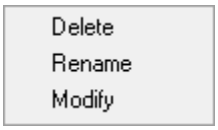
Modify
Opens the Modify Planar Rigid Wall DEZ where you can change the parameters. For more information regarding the options available, see the Define Planar Rigid Wall page.

Single Surface Contacts

Single Surface Contact constraints are available only for the LS-DYNA solver.

You may set Single Surface Contact constraints using the **Define Single Surface Contact** option on the **Constraints** tab, or by right-clicking **Single Surface Contacts** in the Display tree. See the Define Single Surface Contact page for information on the Single Surface Contact options.

Right-clicking on the name of a Single Surface Contact in the Display tree, will give you the following options.



Delete
Removes the selected single surface contact.

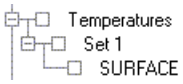
Rename
Opens a dialog box to specify a new name for the single surface contact.

Modify
Opens the **Modify Single Surface Contact** DEZ where you can change the parameters. For more information regarding the options available, see the Define Single Surface Contact page.

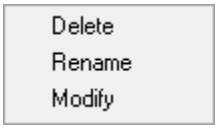
Temperatures

Temperature boundary conditions defined on points, curves or surfaces will appear in the Display Tree as shown below.

Figure 166: Temperatures Tree



Right click on a defined temperature to choose from the following options.



Delete
Deletes the selected temperature boundary condition.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rename
Opens a dialog box to specify a new name for the selected temperature boundary condition.

Modify
Opens a **Modify Temperature Boundary Condition** DEZ which allows you to adjust the parameters of the boundary condition.

For full information on the options available, see Create Temperature Boundary Condition in the Help viewer.

Right click on **Temperatures** in the Display Tree to choose from the following options.

Show All
Displays all the defined temperature boundary conditions.

Hide All
Hides all the defined temperature boundary conditions.

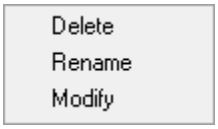
Velocities

You may set Velocity Boundary Conditions using the **Define Initial Velocity** option on the **Constraints** tab, or by right-clicking **Velocities** in the Display tree. See the Define Initial Velocity page for information on the Single Surface Contact options.

When set, velocity boundary conditions will appear in the Display Tree as shown below.

Figure 167: Velocities Tree

Right click on a defined velocity boundary condition set to choose from the following options.



Delete
Deletes the selected velocity boundary condition.

Rename
Opens a dialog box to specify a new name for the selected velocity boundary condition.

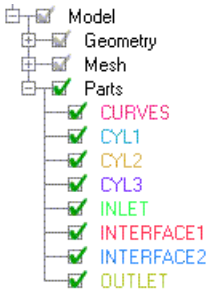
Modify
Opens a **Modify Initial Velocity Parameters** DEZ which allows you to adjust the initial translational and rotational velocities.

For full information on the options available, see Define Initial Velocity in the Help viewer.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parts

Figure 168: Parts Tree



The Parts Display tree contains a list of all the parts in the currently loaded Geometry and Mesh files. If a part is active, then all the mesh and active geometry elements that are associated it will be displayed.

The exception to this rule is if a certain subset is enabled, the entities of that subset are displayed independently of the active parts in the Parts Tree.

Parts Display Options

Right click on **Parts** to view the following display options.

Figure 169: Parts Display Options

The options are described in the following sections:

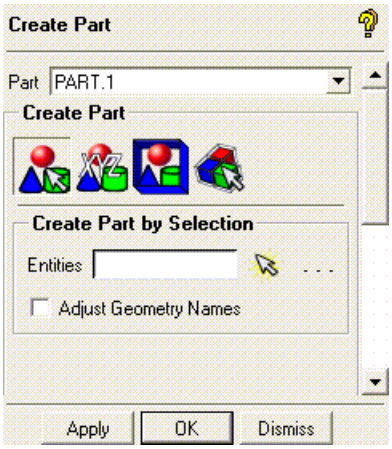
Create Part
Create Assembly
More Part Display Options

Create Part

Parts can be created by four methods.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 170: Create Part Window



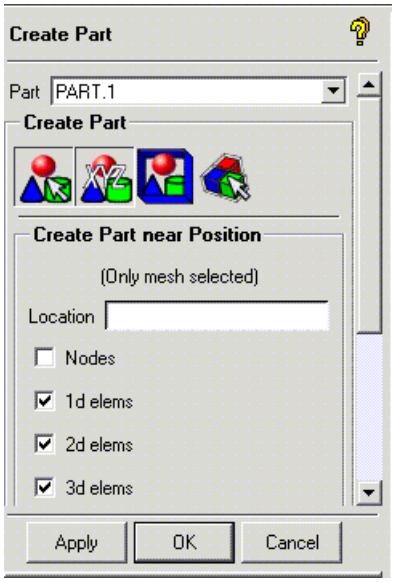
Create Part by Selection

Select geometrical or mesh entities to create a part.

Create Part near Position

Select a position using x, y, and z coordinates to create a part.

Figure 171: Create Part by Near Position window



- **Locations**

  Enter x, y, z coordinates of a specific position.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Create Part in Region

Define a region by specifying the range of x, y, and z coordinates that bound the region.

- **Min and Max Coordinates**

  Enter the minimum and maximum coordinates that represent the region.

- **Include partially enclosed**

  Includes elements partially enclosed in the region specified by the coordinates.

- **Nodes**

  Toggle this **ON** if nodes are to be selected.

- **1d elems**

  Toggle this **ON** if 1d elements (lines and curves) are to be selected.

- **2d elems**

  Toggle this **ON** if 2d elements (surfaces) are to be selected.

- **3d elems**

  Toggle this **ON** if 3d elements (volumes) are to be selected.

- Toggle **OFF** the options that you do not wish to select.

Create Part with Blocks

Select blocks to create a part.

Create Assembly

Assemblies can be used in ANSYS ICEM CFD to help organize data. An assembly is a collection of parts.

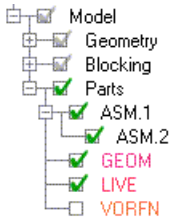
Note

It is important to note that an assembly is treated as a part with the syntax of ASSEMBLY / PART. The "/" character denotes an assembly. In as such, the base level assembly can contain entities, but this is not a recommended practice. Assemblies can contain other assemblies, denoted by the syntax ASSEMBLY1 / ASSEMBLY2 / PART, but it is recommended to keep assembly, sub-assembly, and part names unique.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Assemblies with their constituent parts will appear in the Display tree as shown below.

Figure 172: Parts Tree with Parts and Assemblies



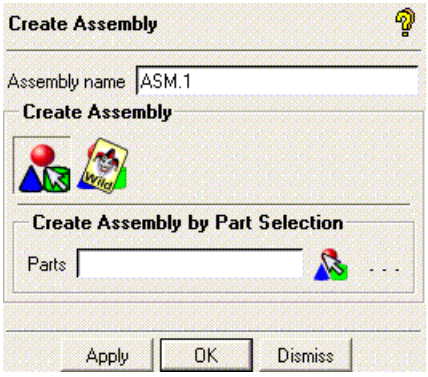
Note

Assembly names appear in black font, and part names appear in non-black colors. If a part color is set to white, a contrast background color will automatically appear in the Display Tree window to make the part names readable.

Assemblies can be created through the following ways:

- Using the Create Part option and entering the syntax ASSEMBLY1 / PART1 as the Part name. Then entities can be selected to be added to PART1 in ASSEMBLY1.
- Parts can also be dragged and dropped into assemblies through the Display Tree. Parts in an assembly can be moved into other assemblies, and entire assemblies can also be added to other assemblies.
- Selecting **Create Assembly** in the Parts Display Options menu will open the **Create Assembly** DEZ.

Figure 173: Create Assembly DEZ



Assembly name

Base level assembly that parts will be added to. If this assembly exists, the selected parts will be added to it. If the assembly does not exist, it will be created.

Create Assembly by Part Selection

This option allows you to select a set of parts to add to an assembly. Parts that are in existing assemblies can also be selected. These parts will be moved to the new base assembly. For example, if the Assembly name is defined as ASSEM2 and a part ASSEM1/PART is selected, the new part will be moved directly under ASSEM2 (named ASSEM2/PART).

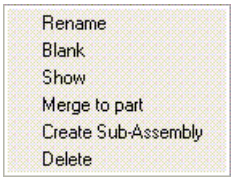
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Assembly by Wildcard

This option allows you to select a set of existing parts to add to an assembly. The **Wildcard text to search** will be used to compare part names and used to define which parts should be moved to the defined Assembly name. For example, if the parts BOX_FRONT, BOX_BACK, BOX_SIDE, INLET, OUTLET exist in a model, then entering the assembly name BOX_ASSEM and entering the Wildcard text BOX will create an assembly BOX_ASSEM containing the three parts BOX_FRONT, BOX_BACK, and BOX_SIDE.

Assembly Display Options

Right-click on any assembly in the Display tree to view the Assembly Display options.



Rename
allows you to rename the selected assembly.

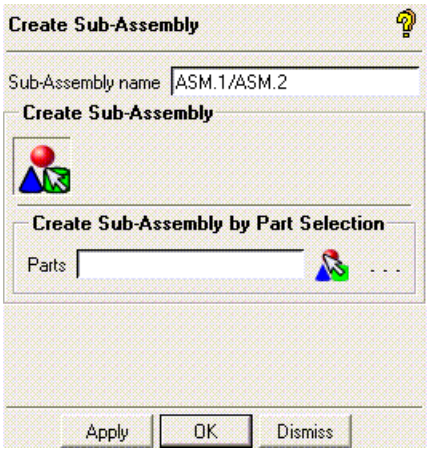
Blank
makes the selected assembly invisible.

Show
displays the selected assembly.

Merge to part
merges the constituent parts with the selected assembly.

Create Sub-Assembly
allows you to create a sub-assembly within an existing assembly from a list of selected parts. Selecting the **Create Sub-Assembly** option will open the **Create Sub-Assembly** DEZ.

Figure 174: Create Sub-Assembly DEZ



Sub-Assembly name
specifies the sub-assembly name. The name also indicates the base assembly within which the sub-assembly is created. For example, the sub-assembly name ASM.1/ASM.2 indicates that ASM.2 is a sub-assembly within ASM.1.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Sub-Assembly by Part Selection

This field allows you to select a set of parts to be added to the sub-assembly. Parts in existing assemblies can also be selected, and will be moved to the new sub-assembly.

> **Note**
>
> Parts can also be moved between assemblies and sub-assemblies by dragging and dropping within the tree.

Delete
allows you to delete the geometry within the selected assembly.

More Part Display Options

Show All
Shows all parts in the display.

Hide All
Hides all parts in the display.

Reverse Blank All
Reverses which parts are blanked so that blanked parts become visible and visible parts are blanked.

> **Tip**
>
> If you have a large number of parts and want to blank most of them, it may be faster to select the parts you want visible, then use **Blank Select** followed by a **Reverse Blank All**.

Expand All
Expands the entire Part Tree.

Collapse All
Collapses the Part Tree.

Blank Selected
Blanks all selected parts.

Restrict Selected
Blanks all parts except selected ones.

Delete Empty Parts
Deletes empty parts.

Expose Component Parts
Exposes the hidden component parts created in ANSYS ICEM CFD and makes them available in the Parts branch of the model tree.

Boundary conditions in ANSYS ICEM CFD are assigned on a per part basis. When loads, constraints, pressures, forces, displacements, etc. are applied on individual entities, the user usually does not want to see them as separate parts. So a hidden component part is created and the necessary boundary condition will be applied to the hidden part. This is usually preferable since it cuts down

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on clutter and these entities can be viewed or adjusted through the **Loads** or **Displacements** branches of the tree. Such hidden parts will be reported when you display the part information (right-click on a part and select **Info** from the part display options).

For example:

```
--------------------------------------------------------------------------------
Info for part GEOM
--------------------------------------------------------------------------------
Geometry Info ----
Part contains 6 surfaces; Total surface area is 6.0
Part contains 12 curves
Part contains 8 points
Bounding box around part is {0 0 0} {1 1 1}
Part contains 2 hidden component part(s):
 GEOM:FORCE0 contains 1 surfaces
 GEOM:TEMP0 contains 1 surfaces
```

The '**:**' in the part name **GEOM:FORCE0** indicates that the part is a hidden component of the part **GEOM**.

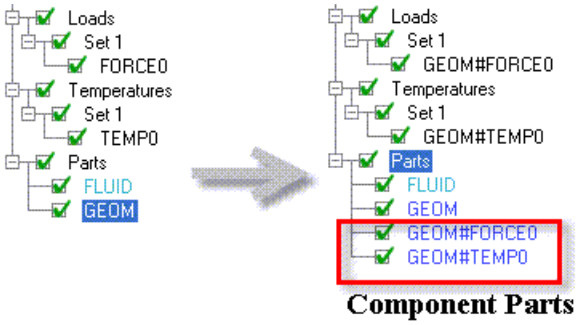
However, for various reasons, some users may occasionally want to expose these hidden component parts as separate parts in the Parts branch of the model tree. This can be done with the **Expose Component Parts** option.

When the **Expose Component Parts** option is used, such parts will be renamed with a '**#**' replacing the '**:**' and they will be available in the Parts Tree.

```
Part GEOM:FORCE0 renamed to GEOM#FORCE0
Part GEOM:TEMP0 renamed to GEOM#TEMP0

2 part(s) renamed
```

Figure 175: Using the Expose Component Parts Option



Note

This operation cannot be undone.

Tip

CATIA V5 and other CAD tools use Component Parts for colors, etc. The hidden component parts are included in the ANSYS ICEM CFD geometry file during conversion. You may find the **Expose Component Parts** option useful to expose the hidden color ID as a part name.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Move Component Parts

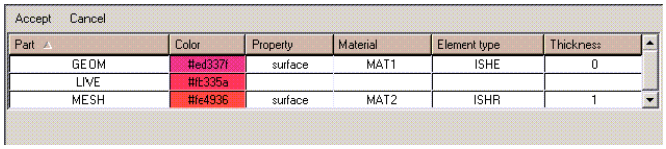
This operation moves the component part into its parent. Unlike **Expose Component Parts** where component parts are renamed, the number of parts in the tree is not increased, resulting in reduced clutter. Following this operation, BCs on the hidden part are ignored. The operation is not reversible.

Edit Attributes

Opens a table of the attributes of all the parts, as shown in the figure below. The part name, color, material, and thickness can be edited directly in the table.

The columns that are grayed out are not applicable for the type of geometry found in that part. Clicking on the column heading will sort the data in ascending or descending order. Columns can be toggled on and off by right-clicking on the headings and selecting the desired options. The column widths can be re-sized by moving the mouse over the column border and dragging it to the desired width. The vertical grab bar at the right of the table can be dragged aside to reveal the Display Window.

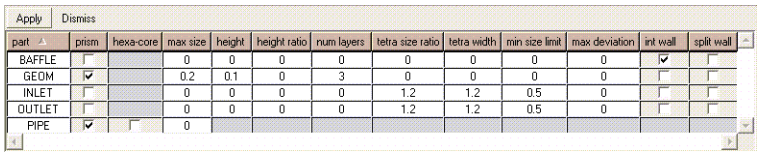
Figure 176: Edit Attributes



Part Mesh Setup

Opens a table of the Part Mesh Setup options. This form is more efficient for large numbers of parts than the Mesh > Part Mesh Setup table.

Figure 177: Part Mesh Setup



The columns that are grayed out are not applicable for the type of geometry found in that part. Clicking on the column heading will sort the data in ascending or descending order. Columns can be toggled on and off by right-clicking on the headings and selecting the desired options. The column widths can be resized by moving the mouse over the column border and dragging it to the desired width. The vertical grab bar at the right of the table can be dragged aside to reveal the Display Window.

"Good" Colors

Automatically selects new colors for all Parts, trying to pick ones that are easy to distinguish from the others in use. Normally, the colors are assigned by an algorithm that employs the parts name, thus, the same family in two different projects will have identical colors. It is not, however, always possible to make all colors distinguishable from the others that are currently in use and the **Good Colors** option provides an alternative.

Color By Material

Assigns colors according to material instead of parts.

Individual Part Display Options

Right-click on any part in the Display tree to view the following display options:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Info

Reports information about the selected part in the message window. The geometry information reported includes the number of surfaces, curves, and points contained in the part, the surface area, and the bounding box extents. The mesh information reported includes the number and the types of elements, the mesh area, and the bounding box extents. The boundary condition information and information about hidden component parts (see Expose Component Parts (p. 185)) will also be reported. For example:

```
--------------------------------------------------------------------------------
Info for part GEOM
--------------------------------------------------------------------------------
Geometry Info ----
Part contains 6 surfaces; Total surface area is 6.0
Part contains 12 curves
Part contains 8 points
Bounding box around part is {0 0 0} {1 1 1}
Part contains 2 hidden component part(s):
 GEOM:FORCE0 contains 1 surfaces
 GEOM:TEMP0 contains 1 surfaces

Mesh Info ----
Part contains 1548 element(s)
Part contains types LINE_2 (0) NODE (0) TRI_3 (0)
Mesh area of part is 6
Bounding box around part is {0 0 0} {1 1 1}
Part GEOM contains 2 hidden component parts:
 GEOM:FORCE0 contains 232 elements of type(s) TRI_3
 GEOM:TEMP0 contains 229 elements of type(s) TRI_3

Boundary Condition Info ----
Part GEOM:FORCE0 contains bc type(s) FORCE MOMENT
Part GEOM:TEMP0 contains bc type(s) TEMP

--------------------------------------------------------------------------------
```

Blank

Blanks the selected part in the display.

Show

Shows the selected part in the display.

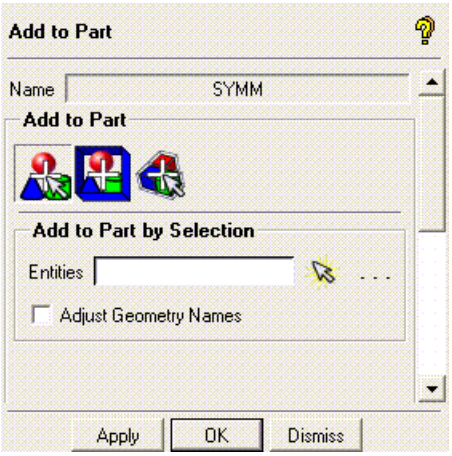
Add to Part

Allows you to add entities to an existing part.

Add to Part by Selection

Allows you to add geometry entities to the selected part. Click (**Select entities**) and select the entities to be added to the part.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

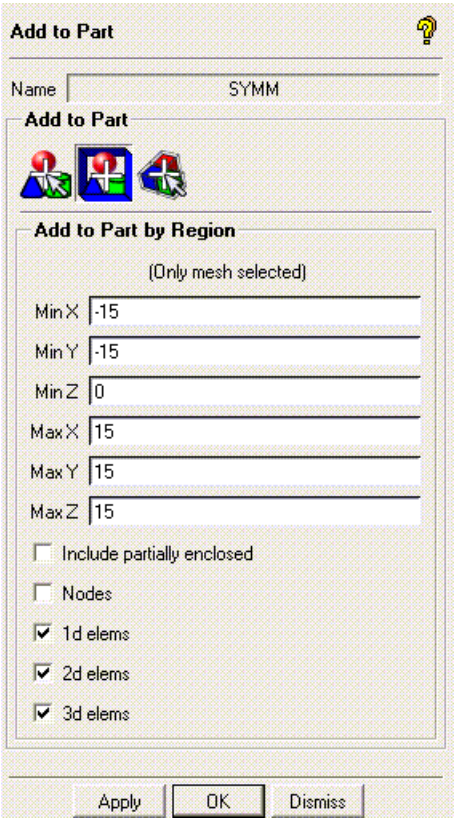


Adjust Geometry Names

Adjusts the name(s) of the entities added to the part based on the original part selected. This option is disabled by default, implying that the original entity names will be retained.

Add to Part by Region

Allows you to add mesh entities within the defined region to the selected part.



Min X, Min Y, Min Z, Max X, Max Y, Max Z

Define the extents of the region.

Include partially enclosed

Allows you to include elements that are partially enclosed by the defined region.

Nodes

Allows you to add nodes within the defined region to the selected part.

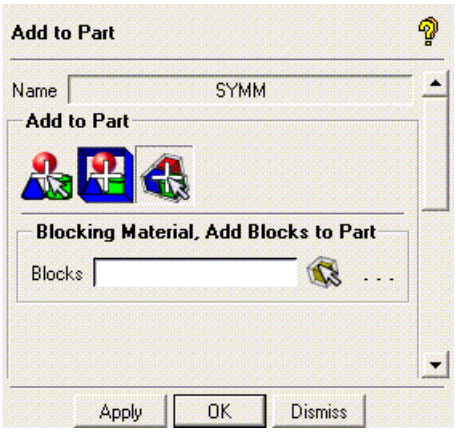
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1d elems
Allows you to add 1D elements within the defined region to the selected part.

2d elems
Allows you to add 2D elements within the defined region to the selected part.

Blocking Material, Add Blocks to Part

Allows you to add blocks to the selected part. Click (**Select block(s)**) and select the blocks to be added to the part.



Break by connectivity
Creates a sub-part comprising all the surfaces from the selected part. This option is valid only for parts containing surfaces. The following additional options will be made available for the original part:

Merge to part
Merges the sub-part comprising the surfaces with the original part.

Create Sub-Assembly
Allows you to create a sub-assembly within the part from a list of selected parts. Selecting the **Create Sub-Assembly** option will open the **Create Sub-Assembly** DEZ (see Create Sub-Assembly (p. 184) for details).

Change Color
Allows you to change the part color to one of your choice.

Rename
Allows you to rename the part.

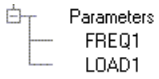
Delete
Deletes the part.

Parameters

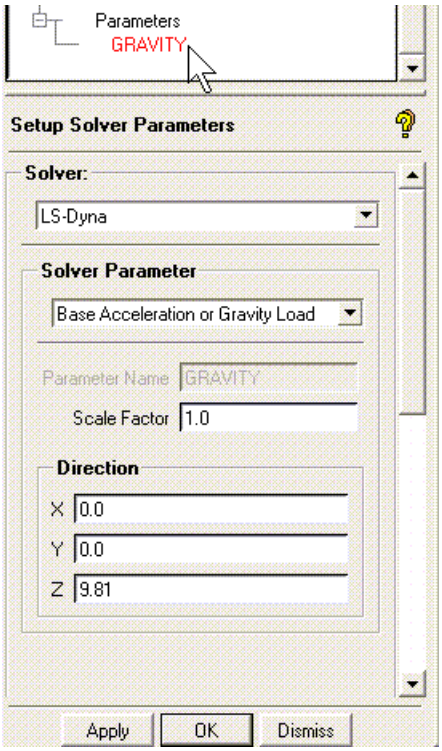
Different solver parameters can be defined to perform different types of analysis. To define a parameter, go to **Solve Options –> Setup Solver Parameters**. See Setup Solver Parameters (p. 601) for details. To edit a parameter, double left-click on its branch in the tree.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 178: Parameters Tree



If gravity is defined, either within the boundary conditions in ANSYS ICEM CFD or in an imported file, it will appear in the Display Tree for editing.

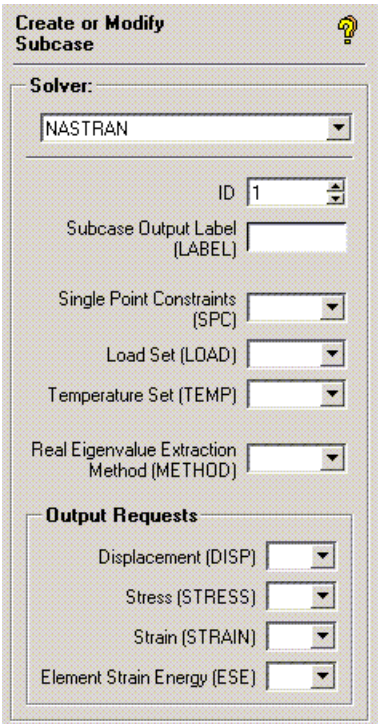


Subcases

Different subcases can be defined to allow you to solve a model for two separate load settings. To define a subcase, go to the **Solve Options** tab, and click on **Set up a Subcase** to open the window shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 179: Create or Modify Subcase Window

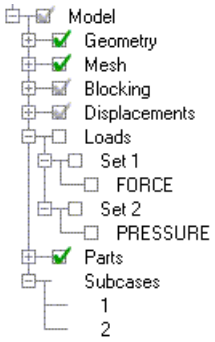


ID

Enter the Subcase ID number.

To define Subcase 2, change **ID** to 2, **LOAD** to **2** and leave **SPC** as **1** and also select **ALL** for DISP, STRESS and ESE as shown in Figure and then click *Apply*. Now you can see the Subcases tree by expanding it in Display Tree which looks as shown in

Figure 180: Display Tree with Two Subcases



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry

Figure 181: Geometry Menu

The **Geometry** tab contains the following options:

Create Point
Create/Modify Curve
Create/Modify Surface
Create Body
Create/Modify Faceted
Repair Geometry
Transform Geometry
Restore Dormant Entities
Delete Point
Delete Curve
Delete Surface
Delete Body
Delete Any Entity

Note

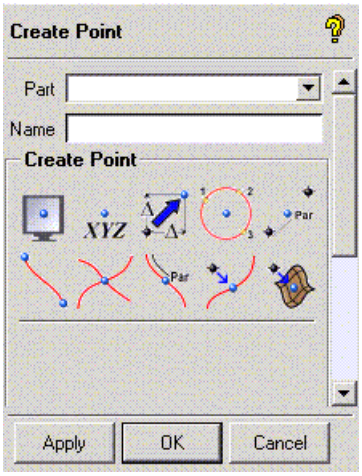
General Naming Convention

Part names and entity names should be less than 64 alphanumeric characters. Names should start with a letter, not a digit. Evaluators (i.e., "+", "-", "/", and "*") should not be used in names because they can be misinterpreted as expressions or as denoting an assembly. Part names are written with all upper-case characters. Entity names are case sensitive.

Create Point

The different options for creating points are shown below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **Part**

  The part name for the newly created point. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the point. If no entity name is given, the name of the point will be the part name with a numeric extension. The default naming convention is described under Settings > Geometry Options.

Note

This name field is very useful when you want to write a script. You can define a specific name for a newly created entity through the **Name** field while writing the script itself rather than the automatically generated name (which could be different in every session depending on the order of creation of the entity).

Screen Select

The **Screen Select** option allows you to select the location of the point by clicking directly on the display window. A temporary bright yellow point will be generated until you press the middle mouse button to complete the selection.

- **Allow work plane selection**

  This option allows you to select locations on any entity as well as in space.

Explicit Coordinates

XYZ The **Explicit Coordinates** option allows you to create a point by specifying the XYZ coordinates, or create multiple points as a function of an equation.

Create 1 point

Enter the X, Y, and Z coordinates of the point to be created.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create multiple points

Allows you to create multiple points defined by functions of one variable (m). Expressions may include: +, –, /, *, ^, ( ), sin( ), cos( ), tan( ), asin( ), acos( ), atan( ), log( ), log10( ), exp( ), sqrt( ), abs( ), distance(pt1, pt2), angle(pt1, pt2, pt3), X(pt1), Y(pt1), Z(pt1). All angles are written in degrees.

m1 m2 ... mn OR m1, mn, incr

There are two possible formats to define the variable m. List format (m1 m2 m3 ... mn) is a simple list of the variable values, without commas. Loop format (m1, mn, incr) is the first value, last value, and increment value, delimited by commas. For example, if the variable m has values 0.1, 0.3, 0.5, and 0.7, this can be represented in both the following ways:

List format: 0.1 0.3 0.5 0.7

Loop format: 0.1, 0.8, 0.2

F(m) –> X

The coordinate X of the points to be created will be calculated as the expression of the specified function of variable m.

F(m)–> Y

The coordinate Y of the points to be created will be calculated as the expression of the specified function of variable m.

F(m)–> Z

The coordinate Z of the points to be created will be calculated as the expression of the specified function of variable m.

Base Point and Delta

The **Base Point and Delta** option allows you to create a point with reference to an existing point. Select the base point and enter in the offset along each axis to describe the location of the new point.

Center of 3 Points/Arc

The **Center of 3 Points** option allows you to select 3 points or locations to define an arc. The center point, or the point that is equidistant from all 3 points, will be created.

Based on 2 Locations

The **Based on 2 Locations** option allows you to create a point based on two locations. Select one of the following two methods:

- **N Points**

  Enter a number (N) of points that you want to create equally spaced between 2 selected locations.

- **Parameters**

  To create a parametric point along a 2-point vector, define the parameter value between 0 and 1, and select the two points that represent the vector.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curve Ends

The **Curve Ends** option allows you to select a curve to create points at the 2 ends of the curve. Specify whether the selected curve is **Bspline** or **Faceted**.

For Bspline curves the following options are available.

Curve(s)
Click on the icon and select the curve(s).

How

- **Both**

  This option creates two points at the ends of the curve. The point at the start of the curve (parameter = 0) will be named by default "pnt.01" and the end point (parameter = 1) will be named "pnt.02".

- **Min or Max coordinates**

  These options will create a point at the location along the curve at the specified minimum or maximum coordinate. For example, if the **xmin** option is selected, the point along the curve with the minimum X coordinate will be created.

  Note

  If xmin = xmax for a curve, xmin will be created at the starting point of the curve and xmax at the end point. This command creates end points and not the peak points of the entire curve.

For **Faceted** curves, the following options are available.

Curve(s)
Click on the icon and select the curve(s).

Angle for curves
Allows you to extract points only for curves with angles greater than the specified angle.

Curve-Curve Intersection

The **Curve-Curve Intersection** option allows you to create a point at the intersection of two curves. Select the intersecting curves.

Gap Tolerance
The distance between the two curves should be less than this value to create an intersection point.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter along a Curve

The **Parameter along a Curve** option allows you to create a parametric point along a curve, select from the following two methods.

- **N Points**

  Enter a number (N) of parametric points that you want to create along a curve.

- **Parameters**

  Define the parameter value between 0 and 1, and select the curve.

Project Point to Curve

The **Project Point to Curve** option allows you to create a point by projecting an existing point on to curve, select the point and curve. The projection will be normal to the curve.

Project Point to Surface

The **Project Point to Surface** option allows you to create point by projecting an existing point on to surface, select the point and surface. The projection will be normal to the surface.

Surface
Select the surface that the point will be projected to.

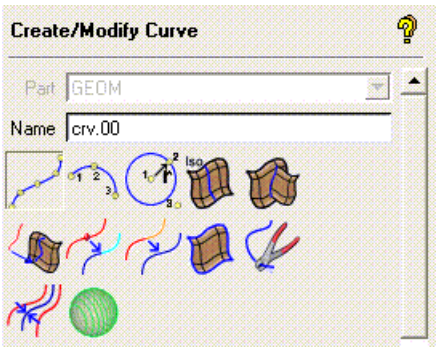
Points
Select the point(s).

Embed point
If enabled, the projected point will become attached to the surface data and a node will be created at the projected point when mesh is generated.

Create/Modify Curve

The different options for creating and modifying curves are shown in Figure 182: Create/Modify Curve Options (p. 197).

Figure 182: Create/Modify Curve Options



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Part**

  The part name for the newly created curve. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the curve. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options.

From Points

The **From Points** option allows you to create a Bspline curve by interpolating through *n* number of points.

From Points
Click on the icon and select any location on the screen to create points that define a curve.

Arc from 3 Points

The **Arc** option allows you to create an arc from three points.

From 3 Points
Creates a Bspline arc through three points.

Center and 2 Points

- **Radius**

  If enabled, the radius will be set as the specified value. If disabled, the radius will be set at the distance between the first two points selected.

- **Keep Center** or **Start/End**

  The **Keep Center** option will take the first point selected as the center, the second point as the first arc beginning, and calculate the arc end from the vector defined by the first point and third point. The radius used will be determined by the **Radius** parameter.

  The **Keep Start/End** option will use the first and second points selected as the arc ends and calculate the center using the **Radius** parameter if specified, or else the average distance between the first and second points, and the first and third points. The plane will be defined by the three selected points.

- **Points**

  Select 3 points.

Circle from Center and 2 Points

The **Circle from Center and 2 Points** option allows you to create a circle of known radius and center point in a plane.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Radius**

  If toggled ON, the radius will be set as the specified value. If toggled OFF, the radius will be set at the distance between the first two points selected.

- **Start angle**

  For a full circle, the start angle is 0. Otherwise, specify the desired start angle.

- **End angle**

  The end angle for a full circle is 360, and for a semi-circle is 180.

- **Points**

  The first point selected is the center point of the circle. The next 2 points selected will be used to define the plane of the circle.

Surface Parameters

The **Surface Parameters** option creates a curve that follows the parametric path of a selected spline surface.

- **Isocurve Methods**

  Select from three methods to create the curve.

  – **Direction on Surface**

    Select two points to indicate the isoparametric curve used to define the new curve.

  – **Point on Edges**

    Select a location on the surface edge to create a curve passing through that location, normal to the edge. You can select any UV parameters, that is, any surface location, but it will automatically use the nearest edge (minimum parametrical distance) from the selected location. A green line will be displayed to indicate the normal direction from the selected edge.

  – **By Parameter**

    Select either the **U** or **V** direction and the parameter value between 0 and 1 to create an isoparametric curve.

Note

If the UV boundary curve data is corrupted, then the following error message may appear: "No active isocurve segments found for this parameter." Use the Geometry > Repair Geometry > Build Topology function to resolve any gaps in the boundary curves.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface-Surface Intersection

The **Surface-Surface Intersection** option allows you to create an isoparametric curve from two intersecting surfaces. You can choose the option to create a faceted curve, or a Bspline curve.

Surfaces
Isoparametric curves will be created for the intersections of the selected surfaces. If **Only Different Parts** is enabled, then curves will be created at the intersections of the selected surfaces from different parts only. If it is disabled, then curves will be created at the intersections of all the selected surfaces. If no surfaces are selected, and **Apply** is pressed, then the operation will be applied to all surfaces.

Parts
This option will apply the function to the selected parts. If **Only Different Parts** is enabled, then curves will be created at the intersections of surfaces from different parts only. If it is disabled, then curves will be created at the intersections of all surfaces within the selected parts. If no parts are selected, and **Apply** is pressed, then the operation will be applied to all parts.

2 sets
This option will apply the function to two sets of selected surfaces. Specify the **Curve Type,** either Bspline or Faceted. Select the sets of surfaces.

Project Curve on Surface

The **Project Curve on Surface** option allows you to create a curve by projecting an existing curve onto a surface. This curve can be used to force nodes and line elements to follow a certain path or to trim the surface as part of geometry creation or repair.

The following methods are available.

Normal to Surface
The selected curves are projected in a normal direction to each selected surface. Multiple curves and surfaces may be selected.

Specify Direction
Select one or more curves and one surface. Specify the direction using the drop-down list. The curves are projected in the specified direction when you click **Apply**.

- **X, Y or Z Direction**

  Projects the curve onto the surface along the specified vector: X direction (1 0 0), Y direction (0 1 0), or Z direction (0 0 1).

- **Screen Normal**

  Projects the curve based on the screen orientation of the model. The curve is projected in a normal direction to the screen.

- **Along a Vector**

  Projects the curve to the surface along a vector. Define the vector by selecting two screen locations.

- **Keep original**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When enabled, the original curve is kept. When disabled, the original curve is deleted.

- **Trim surfaces**

  When enabled, the projected curve will trim the surface.

Segment Curve

The **Segment Curve** option allows you to segment an existing curve into new curves. Select from the following four methods to segment a curve.

- **Segment by point**

  Select the point at which the curve is to be segmented.

- **Segment by curve**

  Select the curve and the location on the curve. The curve will be split at the nearest point of intersection with another curve.

- **Segment by plane**

  Define a plane to segment the curve by selecting an axis normal to the plane and a point on the plane, or a vector normal to the plane and two points on the plane.

- **Segment faceted by connectivity**

  To split two faceted curves that are merged together.

- **Segment faceted by angle**

  To split a faceted curve by a specified angle value.

Concatenate/Reapproximate Curves

The **Concatenate/Reapproximate Curves** option merges two or more existing curves into one new curve. The original curve is deleted. Curves can be merged with the following options:

- **Reapproximate Curve**

  Redefines the curve based on a locally set triangulation tolerance instead of the global tolerance. Used to create a smoother curve definition.

- **Concatenate Curve**

  Joins the curves selected that are separated by a gap less than the given tolerance limit.

- **2D Hull**

  Creates a convex 2D hull around the selected curves. The curves are connected into a faceted curve that does not follow any concave shapes.

  – **Tolerance**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The curve approximation tolerance.

– **Max shrink**

The relative shrink wrap tolerance. The value entered should be between 0 and 1. A typical value is 0.2.

– **Four curves**

This option splits the concatenated faceted curve into four edges so that a surface can be created from them.

Extract Curves from Surfaces

The **Extract Curves from Surfaces** option allows you to extract the boundary curves from a surface. Specify whether the selected surface is **Bspline** or **Faceted**.

For Bspline surfaces the following options are available.

Check Topology
If enabled, curves will be created only on surfaces edges that do not have curves attached to them.

Create New
If enabled, a new curve will be created for each surface edge.

For **Faceted** surfaces the following options are available.

Angle for surfaces
Curves will be extracted only from surfaces with angles greater than the specified angle.

Min number of segments
Only curves with the specified minimum number of segments will be extracted.

Which curve segments
Specify whether curves will be extracted from the interior, exterior, or both sides of the surface.

Modify Curves

Curves can be modified with the following options.

- **Reverse direction**

switches the direction of curves so that the curve start and end points are reversed. Left click on a curve and an arrow will indicate its current direction. To reverse the curve direction, confirm by pressing the middle mouse button.

Note

This function is valid only for parametric (Bspline) curves. Faceted curves must first be re-approximated in order to change the direction of the curve.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

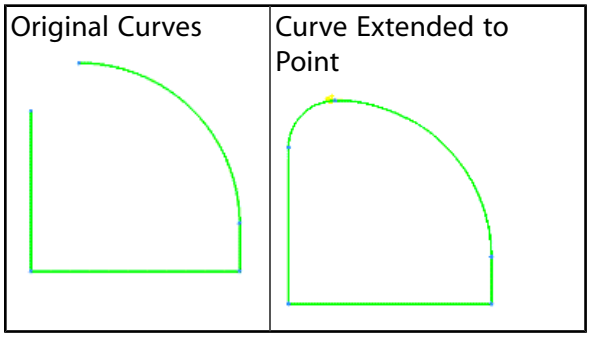
- **Extend**

Extends a curve to the specified point or curve, or extends the curve by a designated length.

Extend to Pnt

Extends the curve to a designated point. When the user selects the edge, the end closest to the selected location is extended. An arrow will indicate which curve end will be extended.

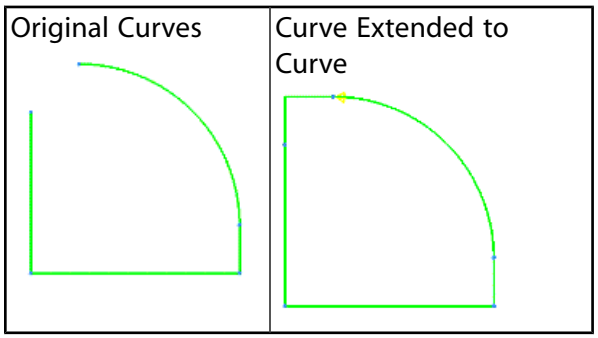
Figure 183: Example of Extend Curve to Pnt



Extend to Crv

Extends the curve to another selected curve. The extension is a straight line tangential with the end of the original curve.

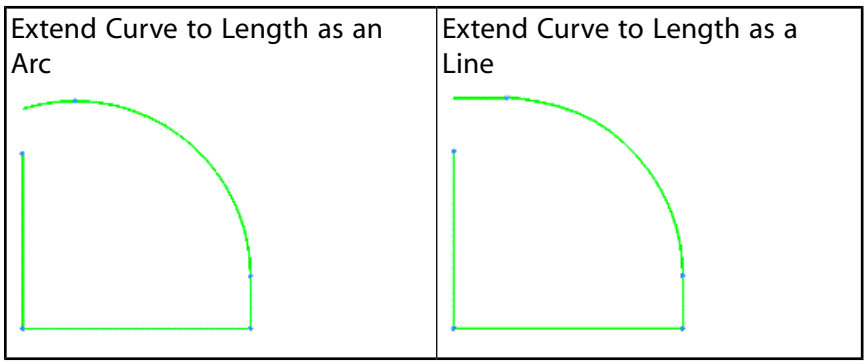
Figure 184: Example of Extend Curve to Crv



Extend to Length

Extends the curve by the specified length. If the curve is not a straight line, it can be extended as an arc or as a line. You can determine which end is extended during the curve selection.

Figure 185: Examples of Extend Curve to Length as Arc and Line



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During extension, a curve will be created in the direction of the extension. The original and extension curves will then be concatenated and the concatenated curve will be re-approximated (using an automatically determined tolerance) and replaced.

Note

The re-approximation will use the default topology tolerance if it is smaller than the automatically determined tolerance value.

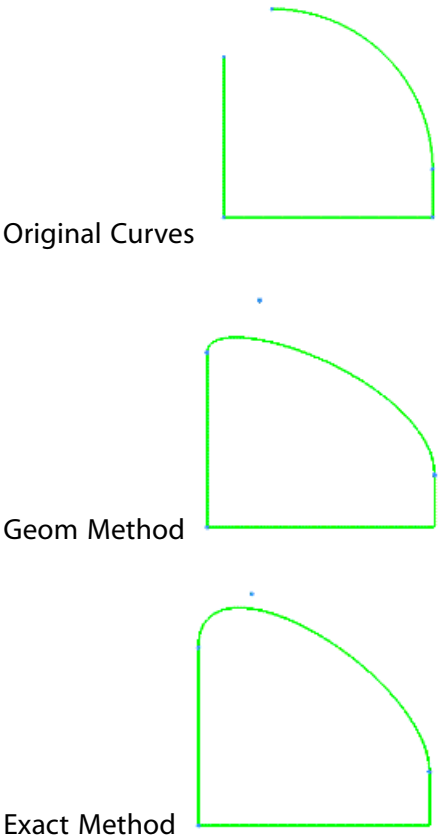
- **Match curves**

Extends curve(s) to match at a point, tangency and/or radius of curvature.

Accuracy

Select the method used to determine the best matching curve. The **Geom** method performs geometric matching with internally set factors. The **Exact** method matches the end point, tangency, and/or radius of curvature as closely as possible.

Figure 186: Example of Match Curves, Geom and Exact Methods



Curves to Modify

The **Both Curves** option will adjust both curves to match. The **First Curve** option only modifies the first curve selected.

Method

This determines the method of matching the curves. The following methods are available:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

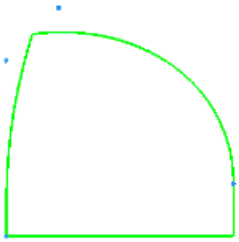
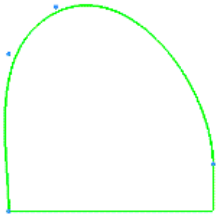
Common end point

Common end point and tangency

Common end point, tangency and radius of curvature

Common end point and radius of curvature

Figure 187: More Examples of Match Curve Methods

Match Curves with Both Curves Option, and Common End Point Method	Match Curves with Both Curves Option, and Common End Point, Tangency, and Radius of Curvature Method
	

Keep Original

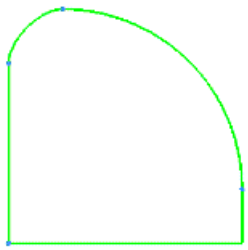
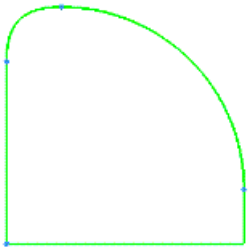
Keeps the original curves and creates new ones to match.

- **Bridge curves**

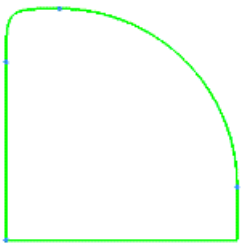
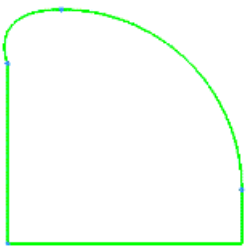
Creates a new curve that connects two existing curves keeping tangency of first two curves. The end that is closer to the location of the cursor when selecting the curves will be the end that is bridged. The **curve magnitude** defines how far to keep the tangency as a fraction of curve length from the start and end point (**First** and **Second Curve**, respectively). The magnitude can range from 0 to 1, where 0 means no tangency is kept for 0 distance, and 1.0 attempts to maintain tangency for the full curve length.

After selecting the curves to be bridged, you can adjust the curve magnitude by text entry or by moving the sliders until the desired shape is achieved.

Figure 188: Examples of Bridge Curves

First Curve Magnitude = 0.30, Second Curve Magnitude = 0.30	First Curve Magnitude = 0.50, Second Curve Magnitude = 0.50
	

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First Curve Magnitude = 0.70, Second Curve Magnitude = 0.70	First Curve Magnitude = 1.00, Second Curve Magnitude = 0.0
	

Notice that for the Curve Magnitudes of 0.70, tangency to both curves cannot be maintained for 70% of the curve lengths, so a compromise is made. For the last example with the First Curve Magnitude of 1.00, tangency to the first curve is attempted for the full length of the bridge, but it does not maintain tangency to the second curve.

Create Midline

The **Create Midline** option creates the midline between two curves or sets of curves.

From 2 Curves
Select two curves and the midline of these curves will be created.

By Pairs
Select two sets of connected curves to create a single midline between them. Each set can contain more than one curve.

Create Section Curves

The **Create Section Curves** option creates new curves at the intersection of selected surfaces with single or multiple planes.

Surfaces
Select the surfaces.

Plane Setup
Specify the planes that intersect with the selected surfaces to create the new curves.

Normal to XYZ Plane
Select the plane normal to the X, Y, or Z axis. The plane will pass through the origin (0, 0, 0).

Start Point / Multiple
This option allows you to define a **Start Point** through which the plane must pass, and allows for multiple planes defined by an offset value.

Delta Offset
To define multiple planes, a Start Point and End Point must be defined. The offset value is defined as a fraction of the distance between the Start and End points. Planes will pass through each offset point, as well as the Start and End points, resulting in multiple section curves. For example, if a Delta Offset value of 0.5 is entered, a plane will pass through the Start Point, the End Point, and the point halfway between the Start and End points.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normal to Three Points
Select 3 points to define a plane. The plane normal to this will be used in the creation of the new curves.

Normal to Existing Curve
Select a curve. The number of sections that is entered will result in equally distributed planes normal to the selected curve at the local parameter. For example, if a curve that is a circle is selected, and 10 sections are specified, then 10 planes normal to the circle, at every 36 degrees, will be used to create the new curves.

Create/Modify Surface

The different options for creating and modifying surfaces are shown in Figure 189: Create/Modify Surface Options (p. 207).

Figure 189: Create/Modify Surface Options

- **Part**

  The part name for the newly created surface. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options.

From Curves

The **From Curves** option allows you to create surfaces from curves. The following options are available for creating surfaces from curves.

From 2-4 Curves
Select two to four curves to form the boundaries of a surface.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tolerance
Adjacent curve ends must be within this tolerance distance in order for boundaries of the surface to be created.

Note

This function automatically closes gaps between boundary curves, using a best fit approximation. For the best results, manually connect curves to form a closed loop that can be specified as the surface boundary.

From Curves
Select any number of curves to form a surface. The curves may be overlapping and unconnected.

From 4 Points
Select four points to define the new surface.

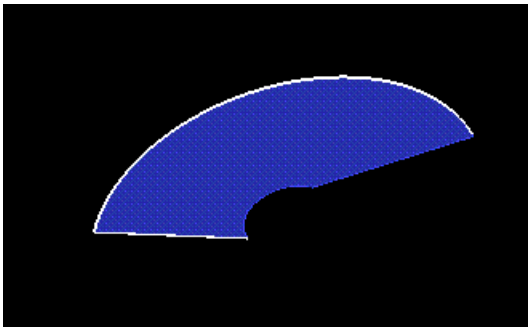
Curve Driven

The **Curve Driven** option creates a surface by sweeping one or more curves along a **Driving** curve. The direction of the **Driven** curves changes with the curvature of the **Driving** curve.

Note

This function provides predictable results for relatively simple driven curves that are either a plane curve or that have a constant curvature. The results may be of lower quality for 3D curves with variable curvature.

Figure 190: Example of Curve Driven Surface



Sweep Surface

The **Sweep Surface** option creates a surface by sweeping a reference curve along a vector (Driving curve). The orientation of the swept curve(s) remain constant, and are swept in the same direction of the vector or Driving curve.

Screen Vector Method
Define the vector by selecting two points, and select the curve(s) to be swept. The vector will determine the size of the resulting surface. For example, for if the two vectors {0 0 0} and {5 3 2} are

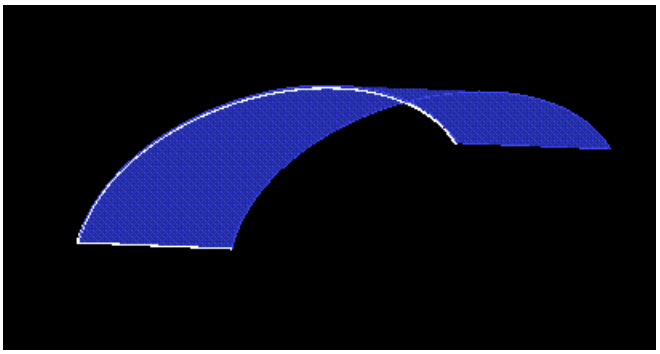
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

entered, the curve will be swept such that a surface of 5 units in the X direction, 3 units in the Y direction, and 2 units in the Z direction will be created.

Driving Curve Method

Select the Driving curve and the curve(s) to be swept. The curve will be swept in the same direction of the Driving curve, from parameter 0 to 1.

Figure 191: Example of Sweep Surface



Surface of Revolution

The **Surface of Revolution** option creates a surface of revolution by revolving a reference curve around an axis with a specified start and end angle. The default start angle is 0, and the default end angle is 360. More than one curve can be chosen to be revolved around the axis.

Loft Surface over Several Curves

The **Loft Surface over Several Curves** option creates a surface by interpolating across two or more curves. The tolerance value determines the degree of approximation. The smaller the tolerance value, the closer the final surface to the input curves.

Offset Surface

The **Offset Surface** option creates a new surface by offsetting an existing surface. Enter the distance to offset the surface normal to the base surface.

Distance

Distance to offset the surface normal to the base surface.

Surfaces

Select the surfaces to be offset.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Midsurface

The **Midsurface** option creates a new surface midway between two existing surfaces or parts. Midsurfaces can be created with the following options.

Note

After midsurfacing complicated models, gaps may sometimes exist between surfaces. To close these gaps, the following operations can be used:

Create/Modify Surface > Extend Surface> Extend Curve to Surface(s)
Create/Modify Surface > Extend Surface> Close Gaps Between Midsurfaced Parts
Repair Geometry > Stitch/Match Edges.

By Parts Method

Select the parts for which midsurfaces are to be created for all paired surfaces. Each part should be a solid part (consisting of surfaces making up a thin solid). The resulting surfaces would represent the midsurfaces for that part.

- **Search Distance**

  The distance within which candidate pairs of surfaces are searched for. For example, if the Search Distance is 10, then all pairs of surfaces within 10 units of distance are selected as candidates for midsurfacing. This value should be slightly higher than the maximum thickness of the entire model. When this parameter is zero, selecting two surfaces will result in finding the midsurface. A nonzero value for the search parameter helps the code to find the midsurface automatically for a large group of surfaces.

- **How**

  This parameter sets the degree by which the operation is interactive.

  – **Quiet**

    This is the default setting, in which the operation is automated.

  – **Confirm**

    After midsurfacing, any surfaces that have not been successfully paired and midsurfaced will be presented. You have the option to delete these surfaces (as they may be side surfaces that should not get midsurfaced), put these surfaces into a subset for further evaluation, or choose **Cancel** to ignore them.

  – **Present/Confirm**

    Will give you the following options for the pair of surfaces for which you want to generate the midsurface. Allows you to view the new midsurface before creating it.

    → **y = compress**

      This option will create the midsurface option that is displayed.

    → **n = no**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This cancels the midsurface operation.

→ **p = partial**

This will present the different options available for non-matching surfaces. The **partial** option will create a midsurface by projecting the smaller surface in between the two surfaces. The **double partial** option will create the midsurface to represent the smallest common surface area of the two selected surfaces. The **non-partial** option will create a midsurface between the entire selected surfaces, not just the overlapping portions.

→ **a = surface 1**

This option will display alternate midsurface options. For example, if one of the original surfaces has a hole, the alternate midsurface options allows you to select the midsurface with or without the hole.

→ **d = delete**

This option deletes the original surfaces.

- **Parts**

  Clicking on the Parts selection icon will open a pop-up window where the parts for midsurfacing can be selected.

- **Keep original**

  Keeps the original surfaces after the midsurface creation.

- **Delete unattached curves and points**

  Deletes unattached curves and points.

- **Create assemblies**

  Creates assemblies of the selected parts for the midsurface creation.

- **Partial**

  For non-matching surfaces, if this option is toggled ON, the midsurface will be created only between the portions that overlap.

  Note

  For the Confirm and Present/Confirm methods, the partial options are presented in the display.

- **Similar pairs only**

  Identifies the matching surfaces from the selected surfaces and generates the midsurface accordingly.

- **Prefer connected pairs**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Is disabled by default. If enabled, connectivity of surfaces over proximity will be used to determine the best surface pairs for creating the midsurface. Also, if there are no connected surfaces, then no midsurfaces will be created. If disabled, then the pairs of surfaces used for midsurfaces will be determined by distance and surface normals.

Note

It is recommended to build topology before using this function.

Note

If **Settings** > **Geometry Options** > **Inherit part name** is set to **Create New**, and **Keep Original** and **Create assemblies** are both disabled, the newly created midsurface will be placed in new part. If both options are enabled, the created midsurface will be placed in the same part, as part of the subset MIDS, and original geometry will be moved to the subset ORIG in the same part.

By Surfaces Method

Any surfaces can be selected to create a midsurface, independent of which part the individual surfaces belong to. All possible midsurfaces will be created for the selected surfaces.

See description of options above.

By Pairs Method

Identifies pairs of connected surfaces via flood fill, re-approximates sides with more than one surface, and then creates a single midsurface between the two sides.

- **Side 1 and 2**

  Specify the different sides of the surface pairs.

- **Keep original**

  Keeps the original surfaces after the midsurface creation.

- **Delete unattached curves and points**

  Deletes unattached curves and points.

- **Partial**

  For non-matching surfaces, if this option is enabled, the midsurface will be created only between the portions that overlap.

Note

For the Confirm and Present/Confirm methods, the partial options are presented in the display.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Segment/Trim Surface

The **Segment/Trim Surface** option allows you to segment a selected CAD surface by a Bspline curve. The selected curve must lie on the surface, and must form an enclosed loop if it is fully contained within the surface boundaries. Surfaces can be segmented by the following four options.

- **By Curves**

  Select the curve(s) to segment the selected surface.

- **By Plane**

  Specify a plane by defining three points on the plane, or by a vector normal to the plane and a point that the plane passes through. The surface will be segmented by this plane.

- **By Connectivity**

  Segments surfaces that are merged by definition but not physically connected.

  Note

  This option applies only to faceted data.

- **By Angle**

  Splits the surface by the value of the angle prescribed.

  Note

  This option applies only to faceted data.

Merge/Reapproximate Surfaces

The **Merge/Reapproximate Surfaces** option allows you to merge two surfaces at their seam. If you select NURBS data surfaces, you will be given the option to convert them into faceted data before merging.

The options for re-approximating surfaces will create one surface from a set of selected surfaces while re-approximating the surface boundaries to a given tolerance. This is useful for several operations, including the following:

1. To clean up surface data by creating a re-approximated surface from multiple segmented surfaces.
2. To clean up more complicated geometric formats into a more simple surface data structure that is easier to manipulate. For example, some CAD systems will represent a four sided surface as a trimmed plane. Re-approximation will create a ruled surface to replace this.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. To convert a faceted surface representation to a Bspline representation if the boundaries of the faceted surface is well defined.

Note

To perform this operation, **Build Topology** is necessary.

There are four options to merge surfaces:

- **Merge Surfaces**

  Merges two surfaces that are separated by a gap within the specified tolerance.

- **Reapproximate selected surface(s)**

  Re-approximates the selected surfaces into one surface.

- **Reapproximate selected surface(s) bounded by curve(s)**

  Re-approximates two adjacent surfaces that are joined by a common curve.

- **Reapproximate selected surface(s) bounded by hull**

  Re-approximates surfaces that are bounded by a hull.

  – **Faceted only**

    Allows you to select faceted surfaces only.

- **Reapproximate each surface**

  Allows you to choose the automated **Quiet** option, or the **Confirm** option that will display the merged surface before creating it.

  – **Faceted only**

    Allows you to select faceted surfaces only.

Untrim Surface

The **Untrim Surface** option is for spline surfaces only. This option untrims or removes any previous segmentation of the selected surface(s).

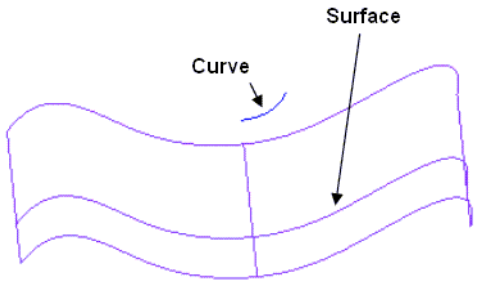
Create Curtain Surface

The **Create Curtain Surface** option creates a surface extruded from a selected curve and projected onto a target surface, normal to the target surface.

Select the initial curve and surface, as shown in the example below. The **Trim surface** option will attempt to trim the set of surfaces selected to form a T-connection.

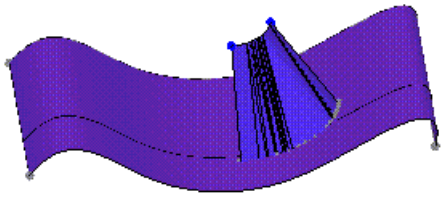
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 192: Select Curve and Surface for Curtain Surface Function



The final surface obtained is shown in Figure 193: Curtain Surface (p. 215).

Figure 193: Curtain Surface



Extend Surface

The following options are available for extending surfaces.

Extend Curve to Surface Method

Extends selected curves to selected surfaces. The following methods and options are available.

- **Closest**

  Projects the selected curve(s) to the selected surface(s), and creates new surface(s) normal to the selected surface(s).

- **Tangential from edge**

  Extends a surface edge to the selected surfaces in a direction normal to the selected edge. The extension follows the curvature of the original surface.

- **Tangential along U/V**

  Extends a surface edge to the selected surfaces in the u or v direction of the original surface.

- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

- **Tolerance**

  This value is the build topology tolerance.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extend Surface at Edge Method

Extends a surface at the selected edge.

- **Surface**

  Select the surface to be extended.

- **Curve**

  Select the edge at which the surface is to be extended.

- **Extension**

  The relative length of the extension. A value of 0.2 defines an extension of 20% of the original surface.

- **Absolute distance**

  If this is toggled ON, then the **Extension** value is taken as an absolute distance instead of a relative factor.

- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

Close gaps between midsurfaced parts

This function requires connected regions of surfaces to be in separate parts. For each part selected, the single edges are checked for their proximity to the surfaces of the other parts. If the proximity is within the specified distance, it will attempt to extend the surfaces in the same way as the Extend Curve to Surface Method.

- **Parts**

  Select the parts that the surfaces belong to.

- **Dist.**

  Enter the maximum gap distance to be closed.

- **Method**

  – **Closest**

    Projects the selected curve(s) to the selected surface(s), and creates new surface(s) normal to the selected surface(s).

  – **Tangential from edge**

    Extends a surface edge to the selected surfaces in a direction normal to the selected edge. The extension follows the curvature of the original surface.

  – **Tangential along U/V**

    Extends a surface edge to the selected surfaces in the u or v direction of the original surface.

- **Preview Vectors**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displays the possible vectors of the surface extension. You can select individual vectors to **Exclude** or **Include**.

- Extension Arrow Size

  Select the vector arrow size.

- **Trim surfaces**

  Attempts to trim the selected surface(s) to form a T-connection.

- **Tolerance**

  This value is the build topology tolerance.

Geometry Simplification

The **Geometry Simplification** option creates a set of faceted surfaces wrapped around the selected geometry, including surfaces, curves, or points. The original geometry is retained.

Convex Hull Method

There are 3 methods to choose from, single hull, interactive split and regular split.

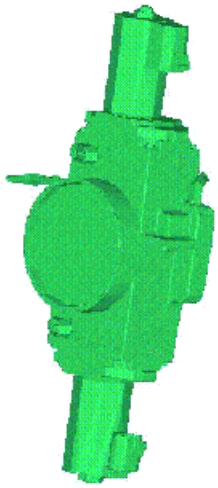
For faceted data, it may be helpful to first merge all surfaces, and then merge all nodes of the concatenated surface within a tolerance. This will reduce the computation time.

There are three different options for the **Convex Hull Method**.

- **Single Hull**

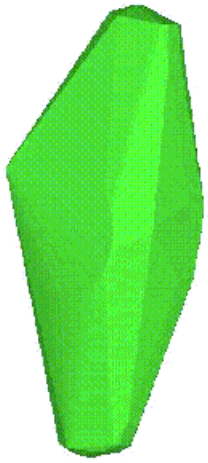
  Creates a crude wrapped surface around the actual selected surfaces as shown in Figure 195: Single Hull Option (p. 218).

  Figure 194: Example Geometry



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

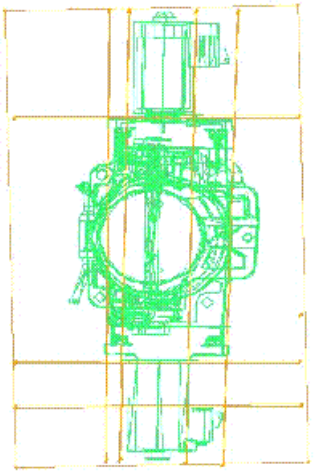
Figure 195: Single Hull Option



- **Interactive Splits**

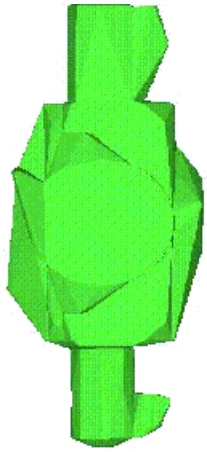
  In this method, you will be prompted to draw lines representing split planes. These planes do not need to be aligned with the global axis.

Figure 196: Split Planes



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 197: Interactive Splits Method



- **Uniform Splits**

  In this method, a uniformly spaced grid of planes is used to segment the actual selected surfaces and a hull is then created around each segment. You must define the number of planes in the X, Y and Z directions. The planes (X, Y, Z axes) are either aligned to the global coordinate system (the **Model** option) or to that of the current **Screen** orientation, with the X-axis horizontal, Y-axis vertical, and Z-axis normal to the screen. A large number of planes will create a three-fold large number of segmented components, which may take a long time to process for complicated geometry. For example, a 10 x 10 x 10 grid will create 1000 components.

Coarsen before Creating Hull
Faceted entities will be coarsened to reduce the triangulation and the processing time.

Tolerance
Edges of the faceted triangles will be merged within the tolerance setting during the coarsening process. The value is an absolute setting.

Remove Interior Faces
If enabled, the interior hull surfaces of segments created by the **Interactive Splits** or **Uniform Splits** methods will be removed. Only the outer shell formed around the selected surfaces will be retained.

Hull for Each Part
Each segment created after a hull is built around a component will be put into a new part.

Cartesian Shrinkwrap

Wraps a complicated geometry using a Cartesian staircase mesh. The mesh is then locally projected and smoothed to get a simplified representation of the geometry.

Note

For the resulting mesh, if surfaces mesh sizes are set, then this will override the global max size value.

The following options are available:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max. cell size
The size of the facets of the shrinkwrap. This should be set to just a little larger than the largest hole you want to ignore.

Num. of smooth iterations
Number of smoothing iterations to be performed.

Surface projection factor
Factor from 0 to 1 that determines how well the shrinkwrap will project to the geometry after the staircase mesh is created.

Part for envelope
Determines the Part that the Shrinkwrap will be created in. You can select an existing part, use the screen select to select a part on the screen, or type in a new part name in the field. The **inherited** option will place the shrink wrapped mesh in the same part as the geometry that it lays on, this is determined by a normal vector ray calculated from each facet.

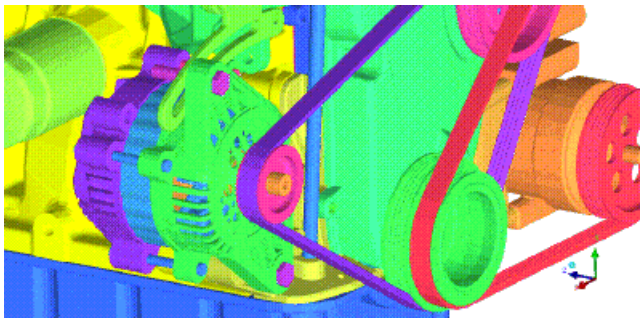
Wrap part by part
Creates an individual shrinkwrap per part.

Active parts only
If enabled, then shrinkwrap will only be applied to the active parts.

Create Geometry
If disabled, the wrap produced is mesh. If enabled, the wrap is automatically converted into a faceted geometry. When used in conjunction with the **inherited** option for **Part for envelope**, the Part names for the new geometry entities will be the inherited names followed by "_WRAP" to differentiate them from the original geometry.

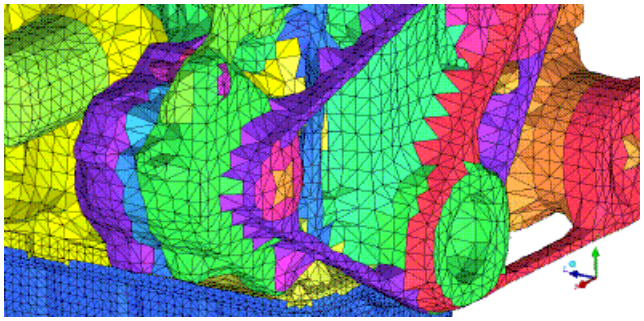
Figure 198: Engine Geometry (p. 220) shows an engine geometry. Such engine geometries may have much more detail than is needed for an under hood analysis and can be simplified. The geometry can be wrapped to close holes, combine parts, and generally simplify the data as shown in Figure 199: Wrapped Engine (p. 221).

Figure 198: Engine Geometry



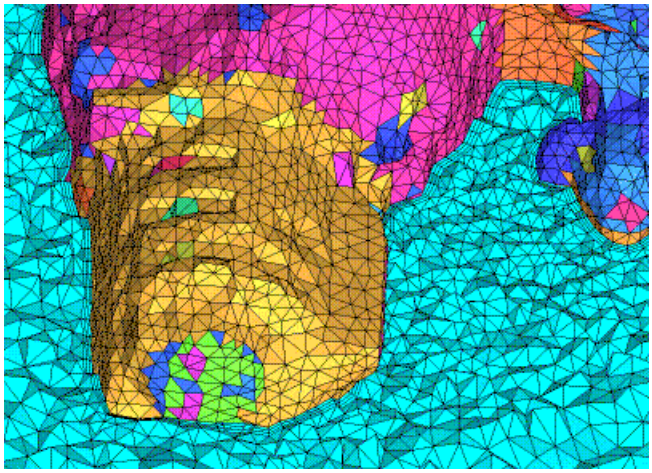
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 199: Wrapped Engine



The wrapped surface mesh can then be used with a bottom-up tetra method to generate a volume mesh or converted to faceted geometry as described.

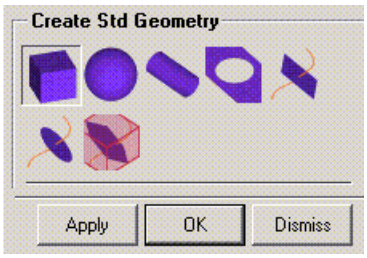
Figure 200: Mesh with Volume Mesh Cut Plane



Standard Shapes

The **Standard Shapes** option creates standard geometrical shapes. The following options are available.

Figure 201: Create Standard Shapes Options



Box

There are two methods to creating a box.

- **Specify**

Define the lengths of the box sides in the X, Y, and Z directions and select the point for the box origin. For example, an **X Y Z size** of "1 1 1" and **Box Origin** "0 0 0" will result in a box with sides of length one, with its origin at the coordinates (0,0,0).

- **Entity bounds**

  Creates a bounding box around the selected entities. The X, Y, and Z lengths of the box can be scaled by the specified factor.

  – **Adjust min/max values**

    If this is enabled, the minimum and maximum coordinate values of the box will be displayed. To create a new box, you can change any of these values and click **Apply**.

Sphere from 2 points

- **Radius**

  If enabled, then the radius length can be defined.

- **Start/End angles**

  The angles (in degrees) are measured from the normal of the center plane of a sphere (the pole). An arc from the Start angle to the End angle will be rotated 360 degrees. For example, a Start angle of 0 and an End angle of 90 will result in the top half of a sphere.

- **Locations**

  You will be prompted to select two locations on the screen. The first is the center of the spherical surface, and the second defines the normal of the center plane (pole). If **Radius** is toggled OFF, then the second point will determine the length of the radius as well.

Cylinder

- **Radius1 and 2**

  Specify the Radius1 and Radius2 of the respective base circles of the cylinder.

- **Base1** and **Base2**

  If enabled, the respective bases circles will be created.

- **Two axis Points**

  Select two points to define the axis of the cylinder.

Drill a Hole

- **Surface**

  Select the surface in which the hole is to be drilled.

- **Remove Hole**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable if you wish to replace a current hole. This will allow you to select a curve (hole) which will be deleted and replaced with the new specified hole. If no curve is selected, the last hole created will be removed.

> **Note**
>
> If **Remove Hole** is enabled, and the selected curve is a feature and not a hole, that curve will still be removed.

- **Radius**

  Enter the radius length.

- **Location**

  Select the point of the center of the hole.

Plane normal to curve

- **Side**

  Enter the length of the side for the uniform surface.

- **Location on curve**

  Select the location on the curve at which the normal plane is to be created.

- **Curve parameter**

  Instead of selecting the location on the curve, the distance along the curve can be specified.

Disc normal to curve

- **Radius**

  Enter the radius length for the disc.

- **Location on curve**

  Select the location on the curve at which the disc is to be created.

- **Curve parameter**

  Instead of selecting the location on the curve, the distance along the curve can be specified.

Trim normal to curve

Creates a plane normal to the selected curve and trims this plane with the selected surface(s). The plane will only be created if the intersection curve of the plane to be created and the selected surfaces is a closed loop.

Surfaces

Select the surface that will be used to trim the newly created plane.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Location on curve

Click on the selection icon and select the location on the curve where the plane will be created. Click on *Apply* to create the plane.

Curve parameter

To specify the location on curve using the curve parameter (between 0 to 1), select any location on the curve, enter the curve parameter, and click *Apply*.

In the example in Figure 202: Surfaces and Curve Selection (p. 224), the top and bottom surfaces of the hemisphere are selected. And the point on the curve is selected for the creation of the plane. In the second figure, note that the created plane's intersection with the selected surfaces creates a closed loop.

Figure 202: Surfaces and Curve Selection

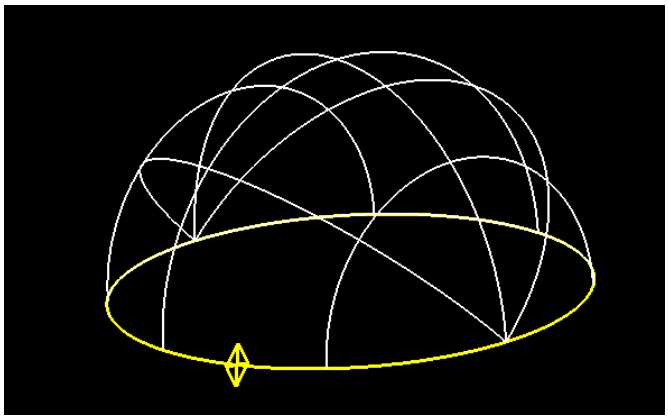
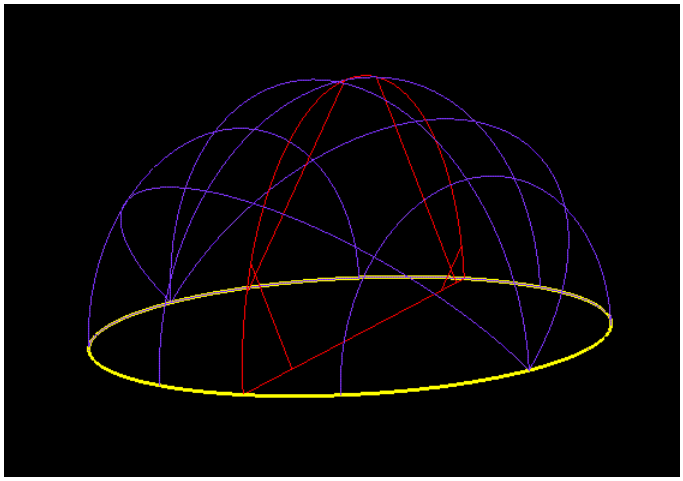


Figure 203: Trim Normal to Curve – Plane Creation



Create Body

Bodies can be created from surfaces that make up a volume. The different options for creating bodies are as follows:

By Topology
By Material Point

- **Part**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The part name for the newly created body. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the body. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options.

By Topology

Bodies can be created by topology either from the **Entire Model** or from **Selected surfaces**. There must not be any gaps in order for a body to be created.

The default part name will be BODY, unless you enter another part name. If no entity name is given, the name of the body to be created will be the part name plus a numeric extension.

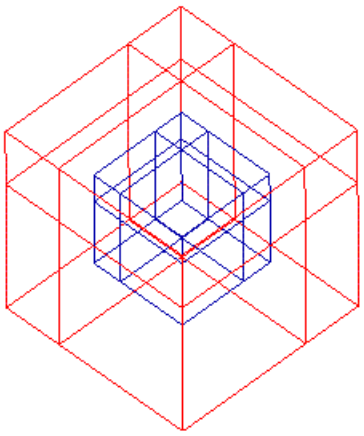
Note

In order to import Tetin files from ANSYS ICEM CFD into the Mechanical application or the DesignModeler application, use the **Entire Model** option to build bodies from the topology first.

Note

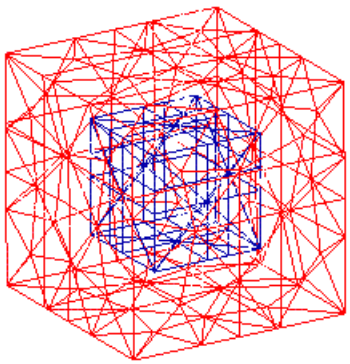
A body will be created for each closed volume. In the example geometry shown below, if the **Entire Model** method is chosen, or if all the surfaces are selected with the **Selected surfaces** method, two bodies will created from the two closed volumes. Each body can then have separate volume meshes.

Figure 204: Create Body by Topology



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 205: Volume Mesh of Separate Bodies



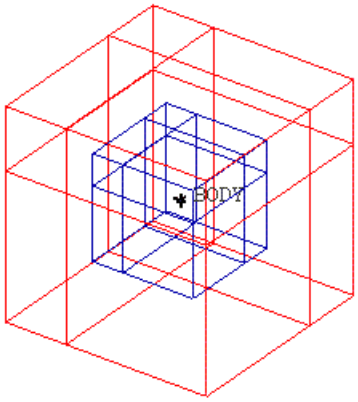
By Material Point

Bodies can also be defined by specifying a Material Point within a closed volume. Material Points can be created at a specified point or at the centroid of two points.

Note

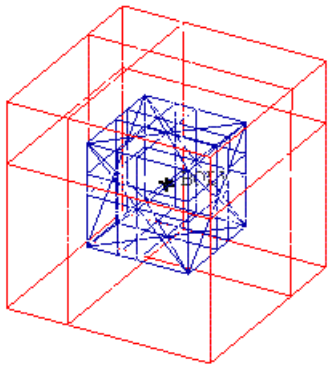
If a material point is specified at the centroid of the cube in the example geometry below, then the body will be defined as inner cube. A volume mesh can be generated for the body as shown below.

Figure 206: Creating Material Point



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 207: Volume Mesh of Defined Body

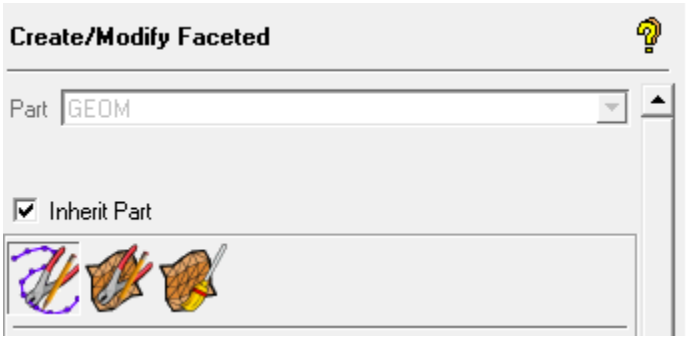


Create/Modify Faceted

The **Create/Modify Faceted** option allows you to create/modify faceted curves and surfaces. Faceted curves and surfaces can be created and modified with the following options.

Create/Edit Faceted Curves
Surfaces
Faceted Cleanup

Figure 208: Creat/Modify Faceted DEZ



- **Part**

  The part name for the newly created entity. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the faceted curve or surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options.

Create/Edit Faceted Curves

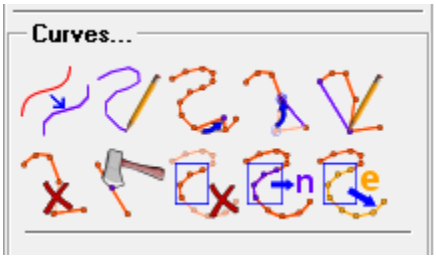
Faceted curves can be created and edited with the following options.

Convert from Bspline
Create Curve
Move Nodes
Merge Nodes

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Segments
Delete Segments
Split Segment
Restrict Segment
Move to New Curve
Move to Existing Curve

Figure 209: Create/Edit Facted Curves Options



Convert from Bspline

The **Convert from Bspline** option converts Bspline curves to faceted curves.

Create Curve

The **Create Curve** option creates a faceted curve from specified locations.

Move Nodes

You can move nodes with any of the following options. After choosing the option, click **Apply**.

Screen
Select the curve, and then click and drag the node to the desired location.

Location
Specify the node and its new location.

Plane
A node can be moved along a plane normal to a specified axis or vector. Define the plane by choosing the normal axis or vector, and click **Apply**. Select the curve, then click and drag the node.

Surface
The node of a curve can be moved along a constraint surface. Select the curve and the surface, then click and drag the node of the curve along the surface.

Line
A node can be moved along a certain line. Specify the line by choosing two points and clicking **Apply**. Select the curve, then click and drag the node along the line.

Offset
Specify the offset in the X, Y, and Z directions to move the node.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge Nodes

A node can be merged to a selected point on another curve. First select the faceted curve, followed by the reference curve. Then the fixed point on the reference curve has to be selected. After that the node selected on the first faceted curve merges with the point selected on the reference curve.

Create Segments

The **Create Segments** option allows you to add segments to an existing curve. First select a curve to which the new segments will be added. Then select points on the screen to create the segments. The selection is in continuation mode: each point selected adds to the segment, until the middle mouse button is pressed. The points are not constrained to existing nodes; rather they are constrained to any location on the visibly displayed geometry.

Delete Segments

The **Delete Segments** option allows you to delete segments from an existing curve. First select the curve to edit, then select the segments to be deleted from that curve.

Split Segment

The **Split Segment** option splits selected segments of the selected curve into two. Split location is at the middle of the element.

Restrict Segment

Select the curve and then select the segment on the curve to be displayed, while the display of all the other segments of the curve will be restricted.

Move to New Curve

The **Move to New Curve** option moves the selected segments to a new curve. Select the curve and then select the segment to move

Move to Existing Curve

The **Move to Existing Curve** option moves the selected segments to a n existing curve. Select the initial and destination curves, then click and drag the curve to the new location.

Surfaces

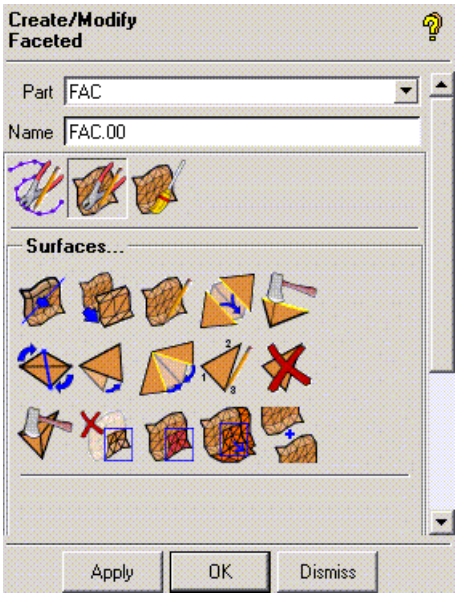
Faceted surfaces can be created and modified with the following options:

Convert from Bspline
Coarsen Surface

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create New Surface
Merge Edges
Split Edge
Swap Edge
Move Nodes
Merge Nodes
Create Triangles
Delete Triangles
Split Triangles
Delete Non-Selected Triangles
Move to New Surface
Move to Existing Surface
Merge Surfaces

Figure 210: Create/Modify Faceted Options



- **Part**

  The part name for the newly created surface. The default naming behavior can be set in Settings > Geometry Options.

- **Name**

  Enter a name for the surface. If no entity name is given, the default naming convention will be followed as described under Settings > Geometry Options.

Convert from Bspline

The **Convert from Bspline** option converts a CAD (Bspline) surface into triangulated surface data. You have the option to delete the old CAD surfaces.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To reconvert the faceted surface into a Bspline surface, see Geometry > Create/Modify Surface > Merge/Reapproximate Surfaces.

Note

Converting a faceted surface to a Bspline surface is a difficult operation and requires the boundaries of the faceted surface to be well defined.

Coarsen Surface

The **Coarsen Surface** option reduces the number of triangles on a faceted surface. This option is useful for reducing the amount of data. After the surface is selected, you will be prompted to select curves and then points in order to maintain the sharp features of the surface. Otherwise, the original surface integrity may not be maintained. The curves and points that have been extracted from the selected surface by one of the following methods should be selected:

Create curve > Extract from surface

Create curve > Build topology

Repair Geometry > Create topology

Create point > Extract from curve

Tolerance

Distance within which to merge nodes of the surface. Default is 1/10th of the Min edge length. The Min edge value is the length of the shortest edge on the selected surface. The tolerance value is a maximum deviation from the original surface triangulation.

Create New Surface

The **Create New Surface** option allows you to create a new faceted surface from three selected points, three locations selected on the screen, or from edges.

The following options are available when creating faceted surfaces using the **From edges** method:

Allow internal edge selection
If enabled, this allows the selection of non-single (internal) edges to create a new surface. Internal edges have more than one adjoining triangle.

Complete edges
If enabled, this allows selection of edges that are not connected.

Merge Edges

The **Merge Edges** option merges one set of edges to another set of edges in faceted surfaces by aligning and merging the edge nodes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Allow internal edge selection

If enabled, this allows the selection of non-single (internal) edges to be merged. Internal edges have more than one adjoining triangle.

Merge tolerance

Merges edges that lie within the specified tolerance of each other.

Merge ends

If enabled, the starting and end nodes of the edges will be merged. If disabled, the starting and end nodes will not be affected.

Split Edge

First select the surface, then the edge to be split. The selected edge and the adjacent elements will be split into two, as shown in the example.

Figure 211: Edge Selected to Split

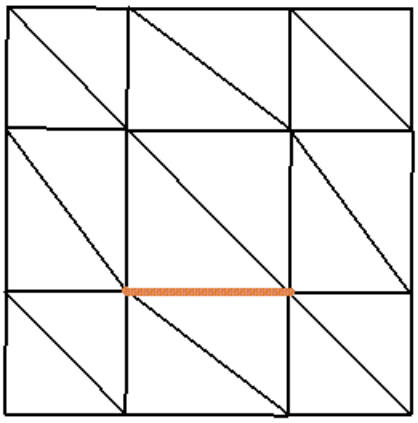
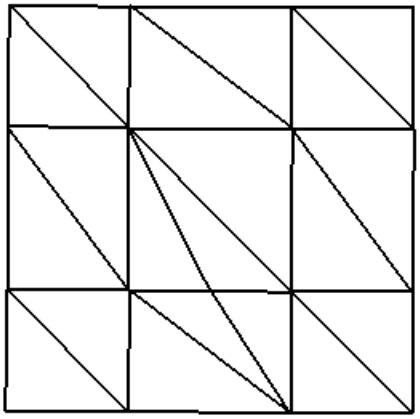


Figure 212: Split Edge



Swap Edge

The **Swap Edge** option swaps edge of two adjacent triangles. The original edge will be replaced by an edge that connects the other two corners of the triangles, as shown in the example.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 213: Edge Selected to Swap

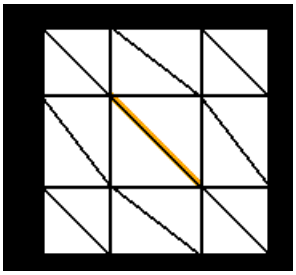
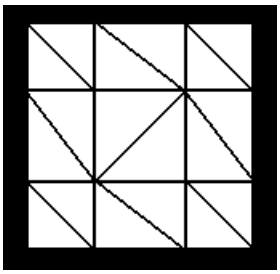


Figure 214: Swapped Edge



Move Nodes

You can move nodes of faceted surfaces with any of the following options. After choosing the option, click **Apply**.

Location
Moves nodes to a selected location on the geometry. First select any location on the geometry, then select the nodes to be moved and press the middle mouse button.

On screen
Moves nodes anywhere on the plane defined by the screen. Select the surface, then click and drag the surface node.

On plane
Moves nodes on a plane intersecting the node and normal to a specified axis or vector. First, select the surface, describe the plane within the menu, and then move the click and drag the node on that plane.

Normal
Define the plane to move the node by typing in the i j k values of the normal of the plane within the box.

Offset XYZ
Specify the offset in the X, Y, and Z directions to move the node.

Line
A node can be moved along a certain line. Specify the line by choosing two points and clicking **Apply**. Select the surface, then click and drag the node along the line.

Merge Nodes

The **Merge Nodes** option merges nodes from one surface with nodes on another surface.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Screen
Merges two or more nodes together. First select the surface with the nodes that are to be merged. Then select the reference surface(s). Select a point on the reference surface to merge the nodes with. Then select the nodes to be merged, and click the middle mouse button to accept. Pressing the middle mouse button twice will exit from the function.

Tolerance
Merges nodes that lie within the specified tolerance of each other. This is useful for reducing data. Other faceted editing functions and repair methods will also be more robust if duplicate nodes within a surface are removed.

Create Triangles

First select a surface to which the new triangles will be added. Then select three points on the screen to create the triangles. The points are not constrained to existing nodes, but can be at any location on the displayed geometry.

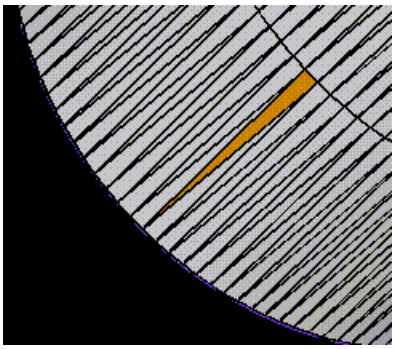
Note

Selecting nodes for creating faceted triangles requires solid or solid/wire display mode for surfaces.

Delete Triangles

Select the surface to edit (the surface facets will be displayed in white), then select the triangles to be deleted from that surface.

Figure 215: Triangle Selected to be Deleted

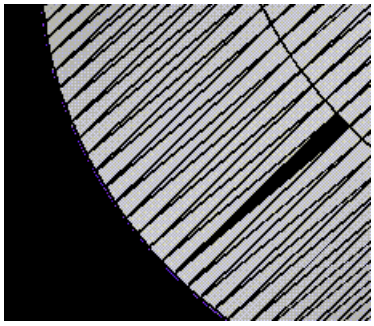


There are two options for deleting triangles:

Delete Selection
Deletes only the selected triangle.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 216: Delete Selection



Keep Selection

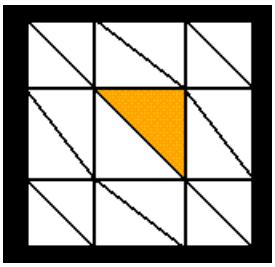
Retains the selected triangle and deletes the rest of the surface.

Figure 217: Keep Selection

Split Triangles

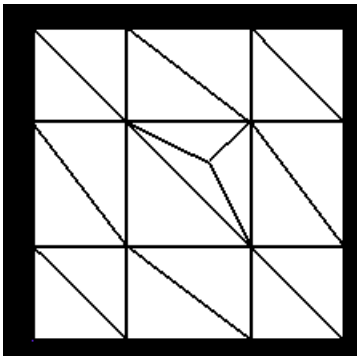
The **Split Triangles** option splits selected triangles into three triangles. The split location is at the centroid of the element.

Figure 218: Triangle Selected to be Split



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 219: Triangle Split



Delete Non-Selected Triangles

The **Delete Non-Selected Triangles** option deletes the triangles that are not selected.

Move to New Surface

The **Move to New Surface** option creates a new surface from the selected triangle(s).

Move to Existing Surface

The **Move to Existing Surface** option moves the elected triangle(s) to an existing surface.

Merge Surfaces

The **Merge Surfaces** option merges two faceted surfaces.

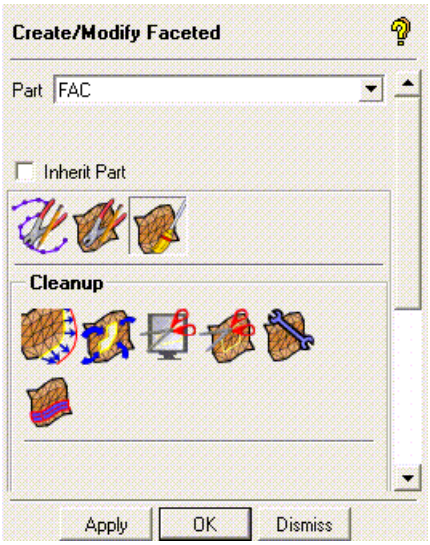
Faceted Cleanup

Faceted surfaces can be cleaned up with the following options:

Align Edge to Curve
Close Faceted Holes
Trim By Screen
Trim By Surface Selection
Repair Surface
Create Character Curve

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 220: Faceted Cleanup Options



Align Edge to Curve

There are three different alignment methods.

Align to Curve

Projects the nodes of the selected surface edge to the nearest location on the selected curve. You will be prompted to select the surface, the curve, and the boundary edge(s) of the selected surface to align to the curve. The nodes attached to the edges will be projected to the nearest location on the curves.

Project Node to Curve

Projects the selected nodes of the selected surface to the nearest location on the selected curve. You will be prompted to select the surface, the curve, and the nodes of the selected surface to align to the curve. This option allows you to select any nodes on the surface in comparison to the **Align to Curve** method, where only boundary edges/nodes can be moved. The nodes selected will be projected to the nearest location on the curves.

Smooth Edges

This option will take a set of boundary edges and smooth out the transition along the edge. You will be prompted to select a surface, and a set of boundary edges of the surface. The software will internally create a temporary Bspline curve through the nodes of the edges using the tolerance value entered for **Smooth Tol**, and project the nodes to this smoothed Bspline. By adjusting the Smooth Tol value you can adjust the transition of the boundary edges.

Close Faceted Holes

The **Close Faceted Holes** option creates new triangles to fill in the gap between a selected set of curves or edges. Select the boundary edges of the hole. The selected curves or edges do not necessarily have to form a closed loop.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trim By Screen

The **Trim by Screen** option cuts a hole through surfaces using points selected from the screen. Select points on the screen to draw the hole on a surface and to cut the hole through all the surfaces below it.

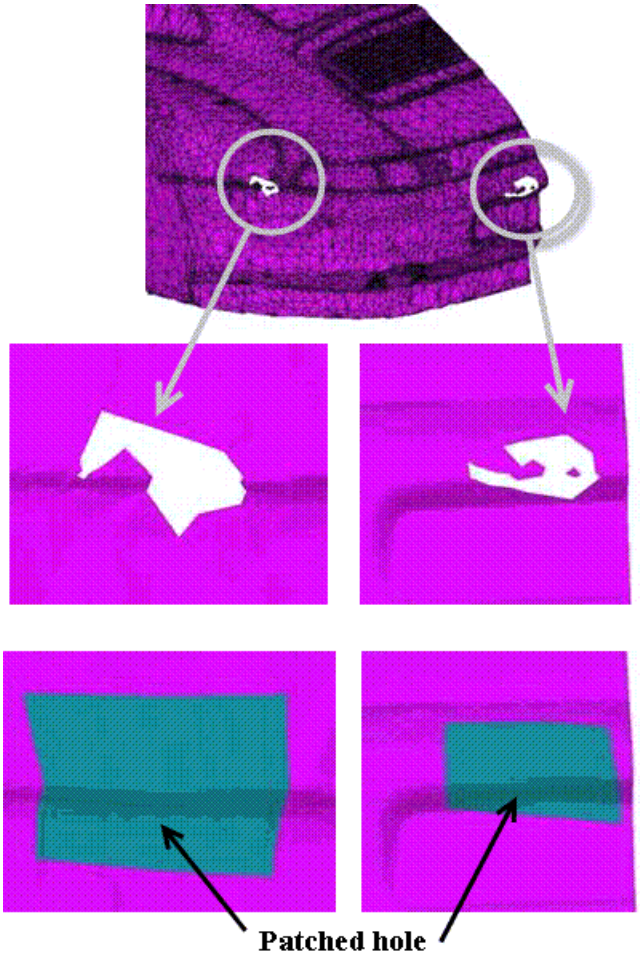
Trim By Surface Selection

The **Trim by Surface Selection** option cuts a hole through surfaces using points on a surface. Select points on the surface to create the hole.

Repair Surface

The **Repair Surface** option creates a Bspline patch to replace an area of the faceted surface. You can define the perimeter of the patch by selecting locations. The patch is projected to the faceted surface. The faceted surface is then cut out and replaced with a Bspline patch. Figure 221: Using the Repair Surface Option (p. 238) shows the use of this option to patch holes in the surface mesh.

Figure 221: Using the Repair Surface Option



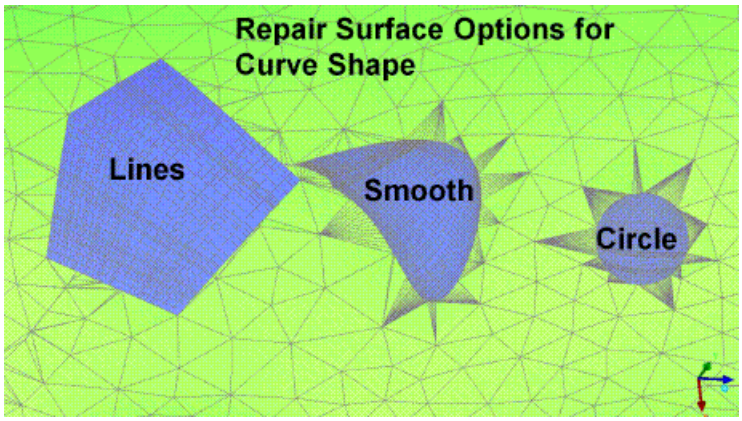
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surf Locations
allows you to select the faceted surface and then select locations on that faceted surface to define the patch to be replaced.

Curve Shape
determines how the surface locations are used to define the patch (see Figure 222: Options for Curve Shape (p. 239)).

- **Lines**: Locations are connected by straight lines.
- **Smooth** : Locations are connected by a smooth curve.
- **Circle** : First three locations are used as three points on a circle. The circular shape is then projected to the faceted surface to determine the final patch.

Figure 222: Options for Curve Shape



Keep original
allows you to keep the original faceted surface patch under the Bspline surface patch.

Create Character Curve

The **Create Character Curve** option creates a Bspline patch to replace an area of the faceted surface where two features join. This can be a sharp, smoothed or arc (fillet) connection. You can define the rough character line and how far on either side of this line should be used to make the patch. An array of faceted surface locations on either side of the character line are determined and smoothed out to create a smoother surface. These are then used to extract an intersection curve to define the theoretical intersection of the character curve. Finally, the faceted surface within the character curve area is replaced with the Bspline surface.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 223: The Create Character Curve Option

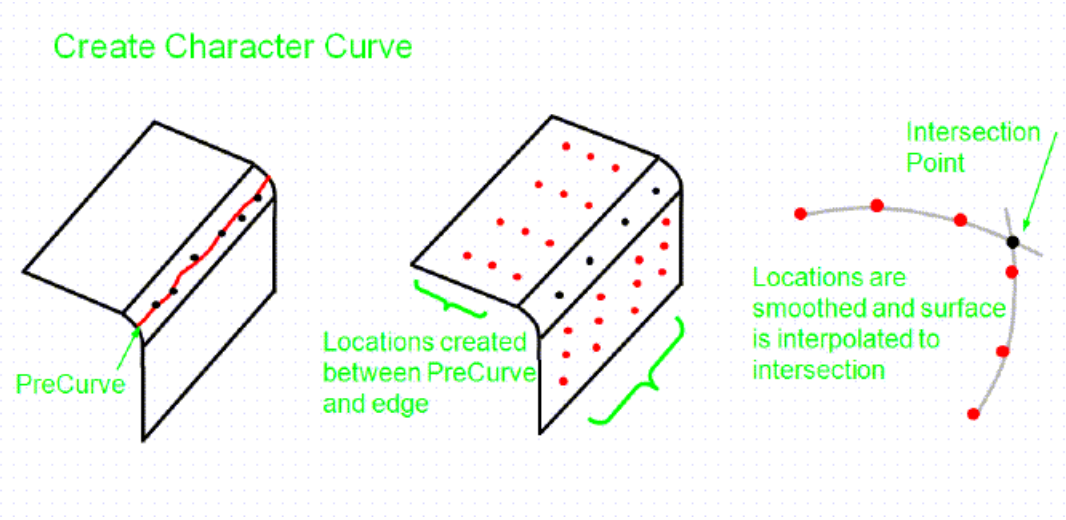
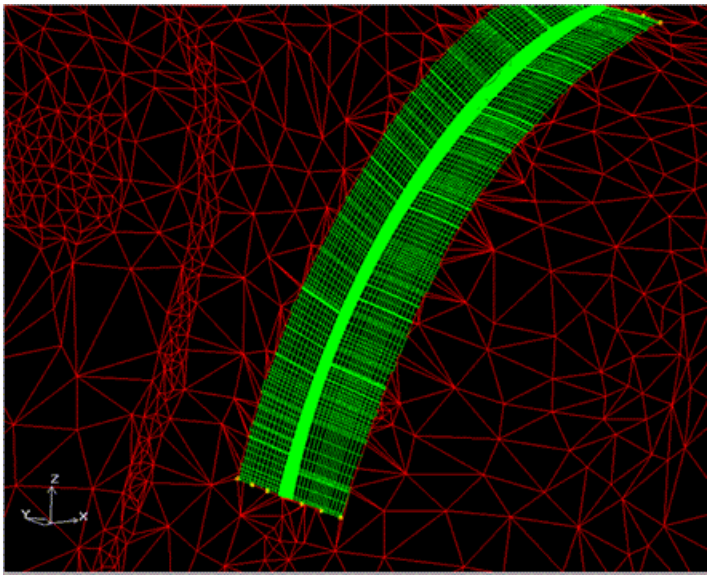

Create Character Curve
PreCurve
Locations created between PreCurve and edge
Intersection Point
Locations are smoothed and surface is interpolated to intersection

Figure 224: Replacing a Faceted Fillet Using the Create Character Curve Option (p. 240) shows the use of this option to replace a faceted fillet with a smoother Bspline fillet.

Figure 224: Replacing a Faceted Fillet Using the Create Character Curve Option



PreCurve
allows you to select locations along the character line you want to capture.

PreSides
allows you to select two locations, one in each plane on either side of the character curve. This determines how far from the character curve the Bspline patch extends in each direction.

Curve Shape
determines how the surface locations are used to define the PreCurve.

- **Lines**: Locations are connected by straight lines.
- **Smooth** : Locations are connected by a smooth curve.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Arc** : First three locations are used as three points on a circle. The arc is then projected to the faceted surface to determine the curve.

Show Presurface
shows a preview of the Bspline surface before applying the function.

Note

This was part of the **Mesh Prototyper** product functionality and the algorithm includes other options which can be scripted for.

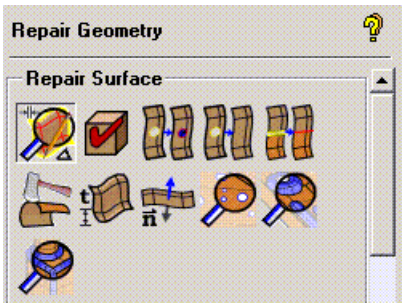
Repair Geometry

The main objective in CAD repair is to detect and close gaps between neighboring surfaces (faces). Typically, the procedure for geometry repair is as follows:

1. Build Topology builds curves and points, which will help to diagnose the model for geometrical problems. The curves will automatically take on colors to show their association to adjacent surfaces.

2. Repair any gaps or holes in the topology.

The options for repairing geometry are outlined in subsequent sections.:
Build Topology
Check Geometry
Close Holes
Remove Holes
Stitch/Match Edges
Split Folded Surfaces
Adjust Varying Thickness
Make Normals Consistent
Feature Detect Bolt Holes
Feature Detect Buttons
Feature Detect Fillets



- **Part**

The part name for newly created curves and points. The default naming behavior can be set in Settings > Geometry Options.

Note

Existing entities will keep their names and parameters.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Build Topology

The **Build Topology** option creates a series of curves and points from surface edges and corners depending on the proximity of the surface edges to each other. If the curves are within a geometric tolerance, they are merged together as one. The curves are then displayed in a specific color to illustrate their connectivity in the surface data, which can be used to determine any gaps or holes in the geometry.

Note

Dormant Entities

Curves and points can be made dormant either by deleting them with the Delete permanently option disabled, through building topology with feature angle and filter points/curves enabled, or through feature detection tools. If a curve or point is dormant, it will not be recreated when building topology, but build topology may make it active. If a curve or point is permanently deleted, it is removed from the database. If the topology is rebuilt, the curve will be recreated. Flood fill, and patch based meshing will also fail across a boundary (curve or point) that has been permanently deleted. Curves and points that are unattached to surfaces or curves will always be permanently deleted, as the sole purpose of dormant curves and points is to maintain connectivity between attached geometry.

Note

Embedded Entities

Points that are within the build topology tolerance from a surface, and curves that are parallel to a surface within the build topology tolerance and not on the boundary of the surface will be embedded into the surface. If a point or curve is embedded or linked to the surface it means that the surface has connectivity to these curves and points. The patch dependent mesher requires this connectivity to include the curve/point as a boundary to the mesher. An embedded curve or point is different than a boundary curve or point in that it is doubly connected to the surface instead of having a single connection like a boundary entity. An embedded curve is connected to the surface twice (on positive and negative sides of the surface), whereas the boundary curves are only attached to the surface on one side.

If the embedded curve is attached to a second surface, the embedded curve becomes a multiple edge, and defines a T-connection. Typical T-connections connect 3 surfaces, but since in the case of an embedded curve the underlying surface is not fully trimmed, the doubly connected edge helps represent the geometry in a more accurate fashion. That is, the connection is defined to the positive and negative side of the base surface and to the boundary of the second surface.

Table 4: Build Topology Colors

Yellow	Single or free edge curves (those adjacent to only one surface)
Red	Double edge curves (curves adjacent to two surface)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blue Multiple edge curves (curves adjacent to three or more surfaces)

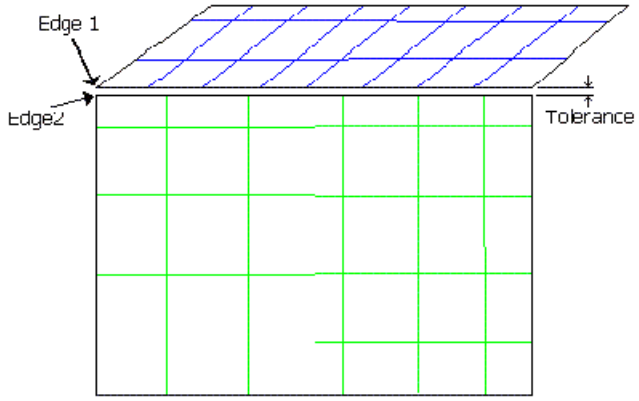
Green Unattached curves (not attached to any surface)

Topology can be built with the following options.

Tolerance

The tolerance is defined in units of the model, and controls how accurately you want to treat surface-to-surface proximity. The tolerance between two surfaces is shown in Figure 225: Tolerance (p. 243).

Figure 225: Tolerance



For any two faces (surfaces) that meet at a common edge (curve), there is typically a finite distance between the two edges. By default, a curve is associated with all the edges of each face. In the figure above, a curve would be associated to edge 1, and a second curve would be associated to edge 2. This topology of the two surfaces would indicate a gap in the model.

Typically, ANSYS ICEM CFD meshers can handle this if the gap is smaller than the proposed element size on the surfaces or curves. Therefore, you would set a tolerance larger than the gap if you are using a large element size. A tolerance smaller than the gap would create yellow curves which could be fixed.

The recommended tolerance is approximately $1/10^{th}$ the size of the average mesh size.

Filter by angle

Filtered curves are made dormant if the surfaces on either side meet at less than the specified angle. Filtered points are made dormant if the curves on either side meet at less than the specified angle. Feature curves and points that meet at greater than the specified angle remain. Since Octree Tetra forces the mesh to respect curves and points, removing unnecessary ones increases the patch independence and mesh quality. Points or curves (even double yellow curves) can also be deleted manually for the same purpose.

Filter points

Enabling this option will filter points by the angle between their respective curves, as long as the angle is not zero.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filter curves
Enabling this option will filter curves by the angle between their respective surfaces, as long as the angle is not zero.

Method
Topology can be built for **All parts**, **Only visible parts**, or by **Selection** of specific entities.

Note

The **All parts** option is different from choosing the **Selection** Method and then **Select all appropriate objects (key = a)**, and will yield a different result. The **All parts** option includes dormant entities, but the **Selection Method** > **Select all appropriate objects** does not include dormant entities.

Part by part
The topology is built part by part.

Single curve cleanup
Enter a value for the **Single Edge Tolerance**. This option will merge single curves only that are within the **Single Edge Tolerance** distance of each other.

Note

This is useful for certain models with small features, where the Build Topology Tolerance must be kept smaller than those features. In such cases, larger gaps with single edges can be cleaned up with this option.

Split surface at T-connections
Surfaces that form a T-connection will be split and trimmed at the common edge. When meshed, the mesh will conform to the common edge.

Split facets at interior curves
Faceted surfaces are trimmed along interior curves that do not span the surface or form a closed loop. When meshed, the mesh will conform to the interior curve.

Join edge curves
Combines edges with the defined angle to concatenate smaller curves into a single curve. This option applies to both Bspline and faceted data.

Delete unattached curves and points
Deletes any green curves and unattached points. Sometimes you may keep these since they could be used as construction curves.

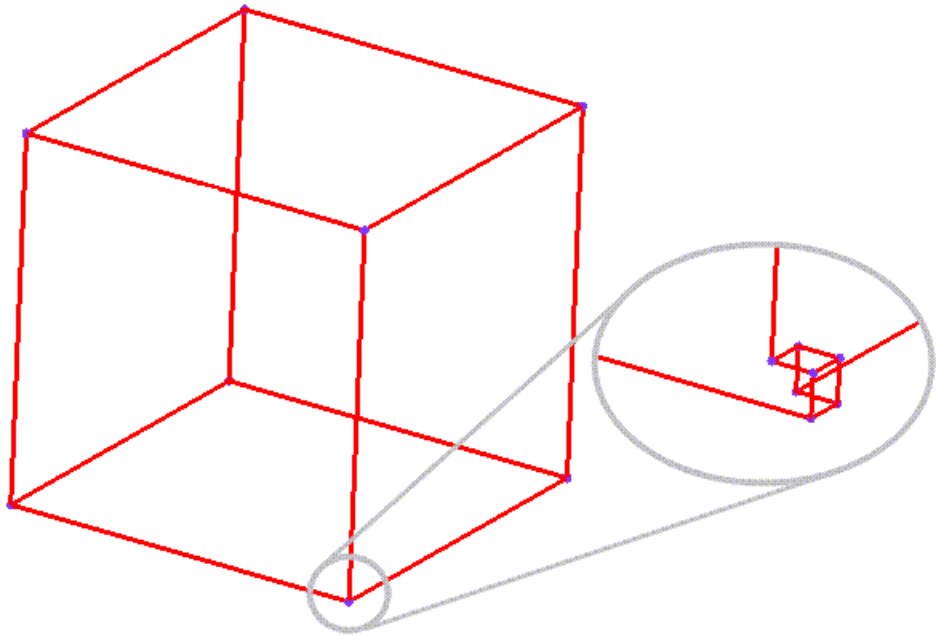
Keep dormant as dormant
Entities that were made dormant by the user will be kept dormant when the topology is built.

Use Local Tolerance
This option enables Build Topology to use a local tolerance, relative to edge length, that prevents merging away small features that are smaller than the global Build Topology tolerance.

This option is useful if a model has a range of scales and the tolerance required for most of the model causes important small features to be corrupted. The model shown in Figure 226: Model with Small Feature (p. 245) has an important feature which is highlighted.

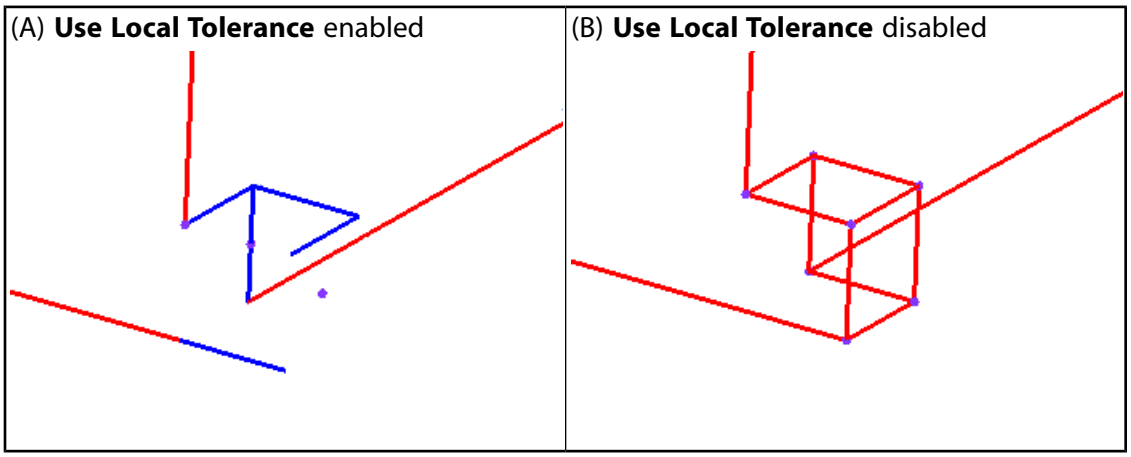
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 226: Model with Small Feature



If the Build Topology tolerance is set to a value larger than the local feature edges, the defined tolerance may be ignored locally by disabling this option.

Figure 227: Use Local Tolerance Option



Check Geometry

The geometry can be checked by all parts, only visible parts, selected parts, or selected surfaces.

The following methods and options can be used to check the geometry.

Results Placement

specifies how the results will be placed. The surfaces that are found can be shown, placed in a specified part, or in a specified subset.

Check Surfaces

- **Edges**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify whether the geometry should be checked for **Single edges** or **Multiple edges**.

- **Curvature/Area**

  If **Curvature** is selected, specify a **High** or **Low** curvature and a **Ref. Angle**. Surfaces with curvatures higher or lower than the reference angle will be found. For example, if a Low curvature of 5 degrees is specified, all surfaces with curvature less than 5 degrees will be found.

  If Area is selected, specify **Large** or **Small**, and the **Ref. Area**. If **Large** is selected, then surfaces that have an area larger than the **Ref. Area** will be found, and if **Small** is selected, then surfaces that have an area smaller than the **Ref. Area** will be found.

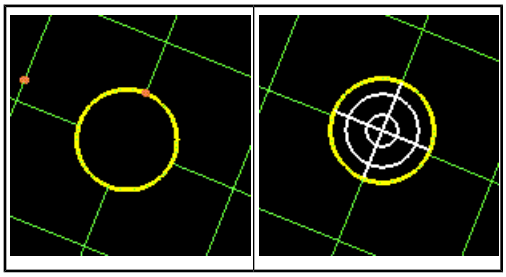
- **Normal**

  This will find all surfaces with a normal within the tolerance angle from the specified axis.

Close Holes

The **Close Holes** option closes holes in the surface by creating a new surface. The necessary condition for closing the hole is that the curve must form a closed loop.

Figure 228: Example of Closed Hole



Curves

Select the curves representing the edges of the hole. After clicking **Apply**, confirm by pressing **Y**.

Multiple Holes

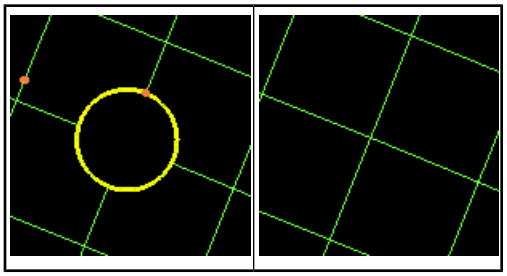
If there is more than one hole, you can select the curves for all the holes at one time. But the holes will be closed individually.

Remove Holes

The **Remove Holes** option removes the hole composed of curves that form a continuous loop. A new surface is not created, as with the **Close Holes** operation. Instead, the trim features adjacent to the selected curves are deleted.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 229: Example of Removed Hole



Stitch/Match Edges

The **Stitch/Match Edges** option stitches or matches edges separated by a gap. Select a pair of curves or multiple pairs of curves to fix. The curve pair will be highlighted in white and will automatically fit to the screen. If multiple pairs have been selected, after each pair is fixed, the next pair will then be highlighted and fit to the screen. The following methods can be used to stitch or match edges:

User Select

After selecting curves, enter the appropriate key to select from any of the methods of stitching or matching edges: (n)o change, (f)ill, (b)lend, (t)rim, (m)atch, (a)uto, or (x) for cancel. Each method is explained below. The **User Select** method is useful when employing a series of different options.

Match

Adjacent surfaces are modified so that one edge matches the other.

Extend/Trim

Recalculates the intersection of the two adjacent surfaces of a gap. Recommended if one or both of the edges appear jagged.

Figure 230: Surfaces Before Extend/Trim

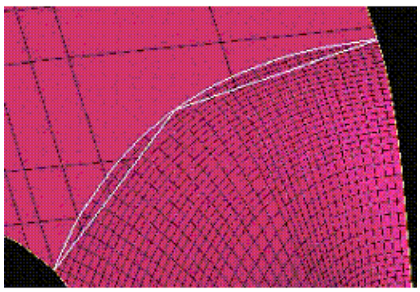
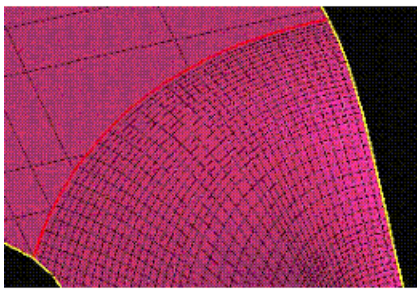


Figure 231: Trimmed Surfaces



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

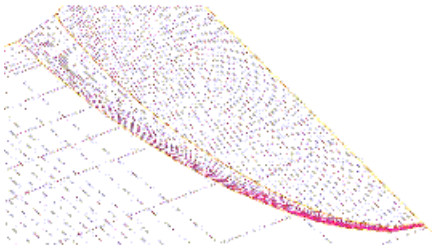
Fill

Creates surfaces to fill in a gap. The surfaces are created between a calculated center point and the selected curves. This is recommended for curve pairs that are more or less coplanar. Any overlap using this method can usually be ignored.

Blend

Creates a ruled spline surface between the curves. Recommended for curve pairs that are not co-planar and that have a high degree of curvature.

Figure 232: Blend method



n = no change

No repair is attempted. If fixing multiple pairs of curves, the next pair will be presented. If only one pair of curves was initially selected, you can select a new set of curves.

a= auto

Automatically determines the best method, **fill, blend, trim** or **match**, to fix the gap or hole.

x= cancel

No repair is attempted and the operation is ended.

p = set/unset partial

Enable or disable for whole or partial fixing of the gap. If this is enabled (prompt will read **Unset partial**), matching will project the ends of the shorter edge normal to the larger edge instead of stretching it to match the entire larger edge; fill or blend will create a surface between the smaller edge and normal projection to the larger edge.

The different parameters for the Stitch/Match Edges operations are described below.

Max Gap Distance

This should be slightly more than the maximum gap between any two surfaces of the entire model. This option is available for the User Select method.

Single curves only

allows you to restrict to single curves only. This option is available for the User Select method.

Partial

Enable or disable for whole or partial fixing of the gap. If this is enabled, (prompt will read **Unset partial**), matching will project the ends of the shorter edge normal to the larger edge instead of stretching it to match the entire larger edge; fill or blend will create a surface between the smaller edge and normal projection to the larger edge. This option is available for the Match, Fill, and Blend methods.

Confirmation

After the repair option is selected, the result will be made visible. You need to give confirmation after viewing the result.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

y = yes
Saves the change.

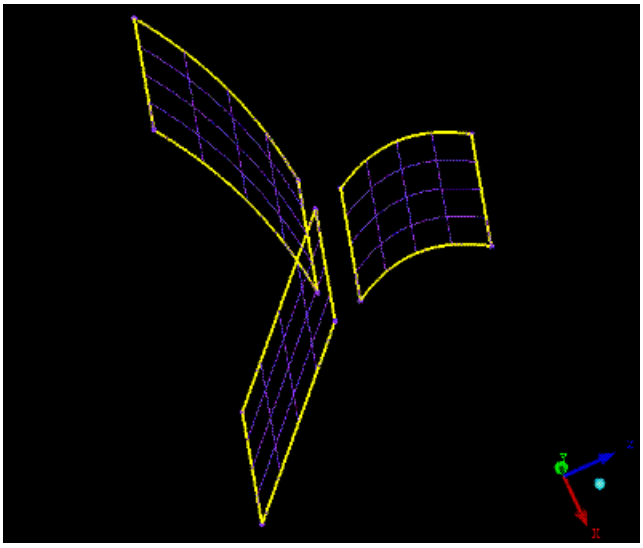
n = no
The change is not saved.

r = retry
To retry repair by selecting a different repair method.

Using Stitch/Match Edges for Y-Junctions

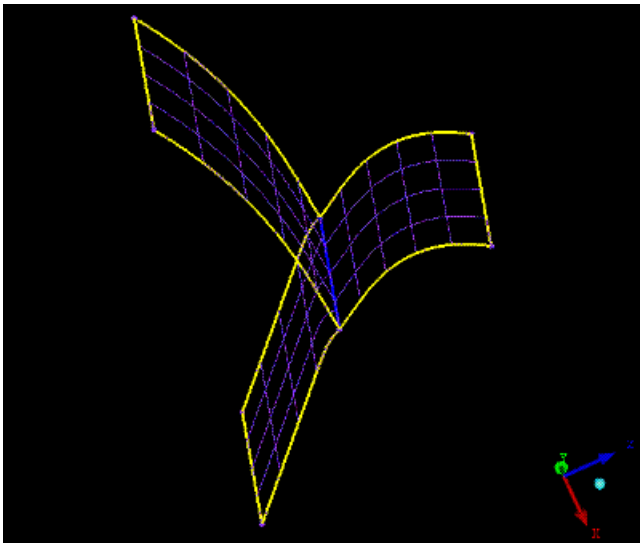
The following example shows how to use Stitch/Match Edges to close gaps in T- or Y-junctions, as shown in the figure below.

Figure 233: Example of Y-Junction with Gaps



First, select the three curves that need to be stitched to close the gap. Select the **Extend/Trim** method. This will trim and stitch two edges together and then automatically connect the third surface. The result is shown below.

Figure 234: Y-Junction with Stitched Edges



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Split Folded Surfaces

Folded surfaces are segmented and new curves created where the surface is folded at an angle greater than the specified **Max angle**. It will help the surface Mesher (Shell) as additional curves are placed on the surface. The **Max angle** option measures from each end of the surface. The first point, from one edge, whose tangent deviates more than specified angle (default is 90 degrees) splits the surface. For a cylinder, this will generate more than one curve.

Adjust Varying Thickness

The **Adjust Varying Thickness** option allows you to assign a thickness to the selected surface using the following methods:

From Solid Method
Specify Corners Method
Find Surfaces Without Thickness Assigned

From Solid Method

This option assigns thickness values to a surface based on the midsurface/solid model.

Midsurface(s)
Select the set of midsurfaces.

Solid surface(s)
Select the set of surfaces on both sides of the midsurface(s).

Max thickness
Limits the thickness assigned to a surface. The thickness values get assigned at certain control points along a surface. Each control point thickness is found by projection normal to the surface in either direction. The sum of these projections (positive or negative) define the thickness. An empty (default) Max thickness value means that there is no thickness limit.

Note

Thickness computations could be inaccurate if the surface data is not very good and a control point projection is not projected correctly. If you see this it would be recommended to try one of the following:

- Improve the surface data.
- Limit the selection of midsurface/solid surfaces to just the local region.
- Reduce the order of the surface to reduce the number of control points.
- Use a maximum thickness to limit the thickness computed.
- Use the **Specify corners** method to define the assigned thicknesses manually.

Order
Defines the number of control points along a surface. For example, Order 4 means 4 x 4 control points along the surface or 16 total control points. For a 4-sided surface, along each edge there would be 4

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interpolation points creating a thickness interpolation surface between the 16 total points along the edges.

Specify Corners Method

This option can be used to assign thickness values to a surface as well as to display the assigned thickness values of a surface.

Surface
Select a surface or set of surfaces. If the thickness values are assigned, they will be displayed. If multiple surfaces are selected, the only value that will be displayed is the value of the first surface selected.

Order
Defines the number of control points along a surface. For example, an order of 4 x 4 control points means that there are 16 total control points along the surface. For a 4-sided surface, along each edge there would be 4 interpolation points creating a thickness interpolation surface between the 16 total points along the edges.

Corner/Middle Points
The thickness values for each specific control point.

Note

There is currently no convenient way to visualize specific control points, so the best way to see the results is by right-clicking Geometry > Surfaces > Show Surface Thickness in the Display Tree. The thickness values can then be assigned in an iterative method.

Find Surfaces Without Thickness Assigned

This option will evaluate a set of selected surfaces and find surfaces that do not have thickness information assigned. If Settings > Geometry Options > Inherit Part Name > Create New is enabled, the surfaces without thickness information will be moved to a new part. If the **Inherit Part Name** option is disabled, then the names of the surfaces found will only be listed in the Message Window.

Note

The Display should be set to Solid mode so that the surfaces which do not have thicknesses assigned will be highlighted in gray, and can be easily recognized.

Part
Enter the name of the new part (or select the default name) or choose an existing part.

Surface
Select the surface or set of surfaces.

Make Normals Consistent

There are two methods available.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make Consistent
Aligns the normals of all the surfaces in the direction of the selected **Reference surface** normal. The check box **Reverse normals** will align the normal of all the surfaces in the opposite direction of the reference surface normal.

Reverse Normal
Reverses the normal of the selected surface.

Feature Detect Bolt Holes

The **Feature Delect Bolt Holes** option detects single edged connected regions in a part whose diameter is between the defined min and max values. When a bolt hole is detected, the following occurs:

- The bolt hole curves are either placed in a subset (if Settings > Geometry Options > Inherited part name is set to **Inherited**), or placed in the defined part name (if Settings > Geometry Options > Inherited part name is set to **Create New**).
- The curves that are marked as bolt holes have the mesh parameter settings modified to the defined Number of quad layers and height ratio (Ring offset ratio). If smart sizing is used, the mesh size will also be adjusted as defined in the note below.

Curves to check
The bolt hole detection will be run on the selected curves. If no curves are selected, the bolt hole detection will be run on the entire model.

Num quad layers
The number of quad layers of uniform height to be grown from the hole. This is equivalent to the Tetra Width mesh curve parameter.

Ring offset ratio
An expansion ratio from the first layer of elements on the curve. This ratio will be multiplied by the element height of the previous layer to define the next layer. You can enter any positive real number. This is equivalent to the Height Ratio mesh parameter.

Min / Max diameter
Bolt holes with diameters within this specified range will be found.

Remove holes below min
If the min diameter is set larger than holes you want to remove, then this option will remove all holes smaller than the **Min diameter**.

Note

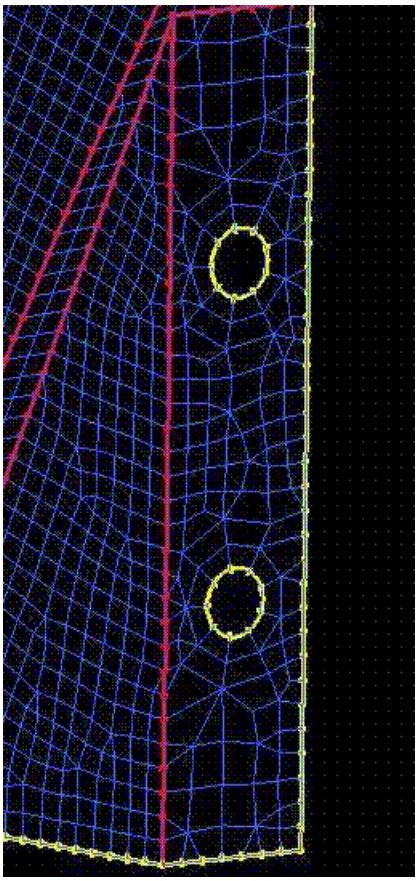
It is recommended to set mesh sizes before using this feature, as the offset layers require the mesh size of the curves to be set properly.

If the mesh size and **Num quad layers** are defined, the smart sizing will try to decrease the mesh size so that the defined mesh size is obtained at the top of the quad layers rather than at the bolt hole. The smart sizing function calculates the ratio of the original mesh size at the outer ring of the quad layers to the new mesh size and reduces the original mesh size by this ratio.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the example below, bolt holes without smart sizing are shown. Uniform mesh spacing was set on all curves prior to bolt hole detection.

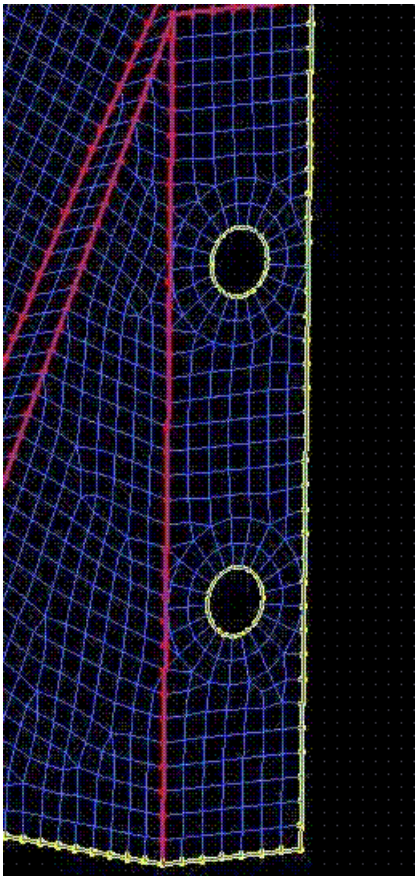
Figure 235: Bolt Hole Without Smart Sizing



An example of bolt holes with smart sizing is shown below, with the same original uniform mesh spacing.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 236: Bolt Hole With Smart Sizing



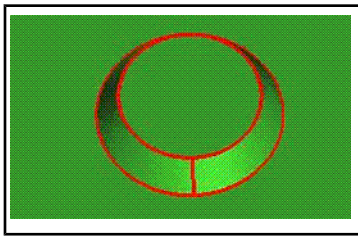
Note

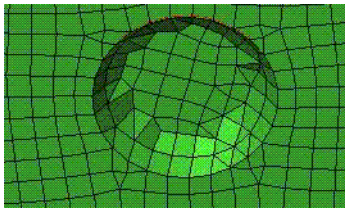
It is important to note that if you run bolt hole detection multiple times with smart sizing, the mesh size will be reduced further each time, as it uses the currently defines mesh parameters to compute what the new size should be. This could lead to excessive refinement, so it is recommended to only run this feature once, or redefine the nominal mesh size on the curves before running bolt holes a second time.

Feature Detect Buttons

The **Feature Detect Buttons** option detects button features and puts them into geometry subsets.

Figure 237: Example of Button Features





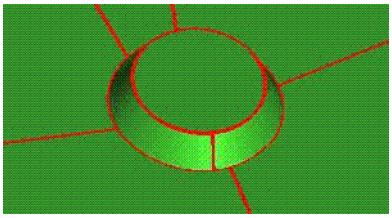
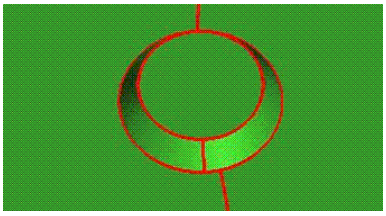
All curves within the button are made dormant so that the meshing of the button is handled in a special way.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are three types of buttons that can be detected: Standard buttons, Fillet buttons and Split buttons. By default the software will look for standard buttons, but there also options to also check for fillet and split buttons.

A fillet button is enclosed by a set of fillets, while a split button is enclosed by two surfaces. See the examples below.

Figure 238: Example of Fillet and Split Buttons

Fillet Button	Split Button
	

Surfaces to check
Select the surfaces that will be checked for buttons.

Fillet buttons
If enabled, the selected surfaces will also be checked for fillet buttons.

Rel. Tolerance
This option is applied to detection of fillet buttons only. If this is disabled, the default tolerance that will be used is 0.001. If this is enabled, a relative tolerance can be specified for button detection.

Fillets are detected if its boundaries are created from arcs. To check whether a curve is an arc, the curve deviation from an arc will be checked using this tolerance.

Split buttons
A split button is a button enclosed by 2 surfaces. If enabled, the selected surfaces will be checked for split buttons as well.

Feature Detect Fillets

The **Feature Detect Fillets** option detects fillet regions and puts them into subsets.

Surfaces to check
Select the surfaces in which to check for fillets.

Min fillet length
Specify the minimum fillet length.

Max fillet length
Specify the maximum fillet length.

Min fillet curvature

Note

The fillet length can be found using the equation:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$L = (180 * R * \theta) / \pi$

where R = radius of the fillet and θ = fillet angle in radians.

All the fillets with lengths between the specified max and min lengths will be placed in the subset.

Specify the fillet curvature in degrees. All the fillets whose curvature is greater than the specified curvature will be placed in the subset.

Relative Tolerance

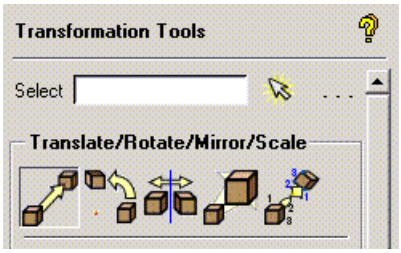
If this is enabled, the default tolerance that will be used is 0.001. If this is enabled, a relative tolerance can be specified for fillet detection.

Fillets are detected if its boundaries are created from arcs. To check whether a curve is an arc, the curve deviation from an arc will be checked using this tolerance.

Transform Geometry

Use transformation tools to change selected entity or entities.

Figure 239: Transformation Tools



The following transformation tools are available:

Translation
Rotation
Mirror Geometry
Scale Geometry
Translate and Rotate

In complex geometry, if only one entity type needs to be transformed, you must display only that entity type and hide others with the display tree options. Clicking the **Copy** option causes the transformation functions to apply to a copy of the selected entities.

By default, all newly created copies will be placed in the same part as the parent entity. To place copies in a new part, use **IncrementParts**, as explained below.

Note

The **Settings > Geometry Options > Inherit part name** toggle does not apply to Transform Geometry options.

Select

Click on the selection icon and select the entities to be transformed.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Translation

There are two methods of defining a translation vector.

Copy
If enabled, a copy of the selected entities will be translated.

Number of copies
Specify the number of copies.

IncrementParts
To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Explicit method
Type the offset distances of the translation for all three directions.

Vector method

- **Through 2 points**

  Select two existing points. Out of these two points the first point selected will represent the tail of the vector while the other will represent the head of the vector.

- **Distance**

  Specify the distance of the translation.

Rotation

The **Rotation** option rotates the selected geometry about an axis.

Copy
If enabled, a copy of the selected entities will be rotated.

Number of copies
Specify the number of copies of the selected entities to be created.

IncrementParts
To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Angle
If this is enabled, you can manually specify the angle of rotation. If disabled, the angle of rotation is automatically set based on the number of copies. For n copies, the angle of rotation is set to 360/(n + 1). For example, for 1 copy, the angle of rotation is 180 degrees, and for 2 copies, it is 120 degrees.

Axis
The axis of rotation can be defined as one of the global axes, or by a vector through two specified points.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Center of Rotation
The center of rotation may be chosen as the Origin, Centroid (of the geometry) or another specified point.

Mirror Geometry

The **Mirror Geometry** option mirrors selected geometry about a plane.

Copy
If enabled, a copy of the selected entities will be mirrored.

IncrementParts
To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

Plane Axis (Normal)
Define the normal of the mirroring plane, either as one of the global axes, or by a vector through two specified points.

Point of Reflection
The point of reflection may be chosen as the Origin, Centroid (of the geometry) or another specified point.

Scale Geometry

The **Scale Geometry** option allows you to scale the selected geometry in all three directions or in any particular direction.

Copy
If enabled, a copy of the selected entities will be scaled.

IncrementParts
To add the newly created copies of entities to a new part, click the IncrementParts icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

X, Y, Z factor
Enter the scale factor for each direction.

Center of Transformation
The center of the scaling transformation may be chosen as the **Origin**, **Centroid** of the geometry, or another **Selected** point.

Translate and Rotate

The **Translate and Rotate** option allows you to translate and rotate the geometry at the same time.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy
If enabled, a copy of the selected entities will be created in the new location.

IncrementParts
To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, GEOM_1, GEOM_2, GEOM_3, etc.

3 points –> 3 points
Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve
Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1) of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

LCS –> LCS
Click **Show LCS** to view any defined coordinate systems. Select two local coordinate systems.

The result will match the origin and the axes of the first to the second LCS.

Restore Dormant Entities

This **Restore Dormant Entities** option restores dormant curves and points.

Dormant curves and points are a fundamental part of ANSYS ICEM CFD feature build topology. When build topology is performed, the software builds connectivity information between surfaces. For example, a surface is attached to a set of curves that is attached to a set of points. This information is needed for flood fill operations, patch based meshing, and many geometry operations that require connectivity information. When a curve is made dormant, the curve is still in the database and can be restored through this feature. Thus the connectivity between the surfaces still exists. Dormant curves and points can also be displayed by right clicking on Curves and Points Display Trees and selecting Show Dormant.

Curves and points can be made dormant either by deleting them with Delete permanently option disabled, through building topology with feature angle and filter points/curves enabled, or through feature detection tools. If a curve or point is dormant, it will not be recreated when building topology, but build topology may make it active. If a curve or point is permanently deleted, it is removed from the database. If the topology is rebuilt, the curve will be recreated. Flood fill, and patch based meshing will also fail across a boundary (curve or point) that has been permanently deleted. Curves and points that are unattached to surfaces or curves will always be permanently deleted, as the sole purpose of dormant curves and points is to maintain connectivity between attached geometry.

Delete Point

The **Delete Point** option deletes selected points.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete unattached
automatically deletes unattached points.

Delete permanently
permanently deletes the selected points and removes them from the database.

Join incident curves
If two curves are attached at a point, and they do not form a sharp angle, then if the point is deleted, this option will concatenate the curve.

Delete All Dormant Points
deletes all dormant points.

Delete Curve

The **Delete Curve** option deletes selected curves.

Delete unattached
automatically deletes unattached curves.

Delete permanently
permanently deletes the selected curves and removes them from the database.

Delete All Dormant Curves
deletes all dormant curves.

Delete Surface

The **Delete Surface** option permanently deletes the selected surfaces and removes them from the database.

Delete Body

The **Delete Body** option permanently deletes the selected bodies and removes them from the database.

Delete Any Entity

The **Delete Any Entity** option deletes selected entities.

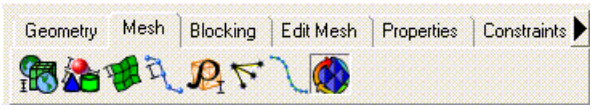
Delete unattached
automatically deletes unattached entities.

Delete permanently
permanently deletes the selected entities and removes them from the database.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh

Figure 240: Mesh Menu



The **Mesh** tab contains the following options:

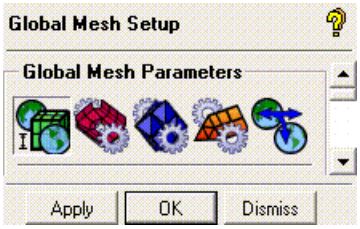
Global Mesh Setup
Part Mesh Setup
Surface Mesh Setup
Curve Mesh Setup
Create Mesh Density
Define Connectors
Mesh Curve
Compute Mesh

Global Mesh Setup

The **Global Mesh Setup** options provide the general and specific meshing algorithm parameters used for the various meshers. The global mesh parameters are stored in the geometry file and help define global controls for the following:

Global Mesh Size
Shell Meshing Parameters
Volume Meshing Parameters
Prism Meshing Parameters
Periodicity Set Up

Figure 241: Global Mesh Setup Parameters



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a comprehensive list of the different meshing parameters and which methods they apply to, see Table 5: Parameters for Surface Meshing Methods (p. 262), Table 6: Parameters for Volume Meshing Methods (p. 263), and Table 7: Parameters for Other Meshing Methods (p. 265).

Table 5: Parameters for Surface Meshing Methods

	Surface Meshing Methods			
Mesh Parameters	**Autoblock**	**Patch Dep.**	**Patch Indep.**	**Shrinkwrap**
Global Parameters				
Max Element*	Y	Y	Y	Y
Min Size Limit*	N	Y	Y	N
Elements in gap	N	N	Y	N
Refinement	N	N	Y	N
Periodicity	N	N	Y	N
Part Meshing Parameters				
Internal Wall	N/A	N/A	N/A	N/A
Thin Wall	N/A	N/A	N/A	N/A
Hexa-core	N/A	N/A	N/A	N/A
Prism setting	N/A	N/A	N/A	N/A
Surface Meshing Parameters				
Ignore Size	Y	Y	N	N
Surface mesh type	N/A	N/A	N/A	N/A
Prism Meshing Parameters				
Growth law	N/A	N/A	N/A	N/A
Initial height	N/A	N/A	N/A	N/A
Height ratio	N/A	N/A	N/A	N/A
Number of layers	N/A	N/A	N/A	N/A
Surface Parameters				
Max size*	Y	Y	Y	Y
Height	N/A	N/A	N/A	N/A
Height ratio	N/A	Y	N/A	N/A
No. of layers	N/A	N/A	N/A	N/A
Tetra width	N/A	N/A	Y	N/A
Tetra size ratio	N/A	N/A	Y	N/A
Min Size Limit*	N	N	Y	N

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Surface Meshing Methods			
Mesh Parameters	**Autoblock**	**Patch Dep.**	**Patch Indep.**	**Shrinkwrap**
Max deviation*	Y	Y	Y	N
Surface mesh type	Y	Y	Y	N/A
Curve Parameters				
Max size*	Y	Y	Y	N
No. of nodes	Y	Y	Y	N
Height	N	Y	N	N
Height ratio	N	Y	N	N
No. of layers	N	Y	N	N
Tetra width	N/A	N/A	Y	N/A
Min Size Limit*	Y	N	Y	N
Max deviation*	Y	N	Y	N
Bunching	N	Y	N	N
Densities*	N	N	Y	N
Connectors	N	Y	Y	N
**Scale Factor is applied*				

Table 6: Parameters for Volume Meshing Methods

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
Global Parameters					
Max Element*	Y	Y	Y	Y	N
Min Size Limit*	Y	N/A	N/A	N/A	N
Elements in gap	Y	N	N	N	N
Refinement	Y	N/A	N/A	N/A	N
Periodicity	Y	N/A	N/A	N/A	N
Part Meshing Parameters					
Internal Wall	Y	N/A	N/A	N/A	N
Thin Wall	Y	N/A	N/A	N/A	N

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
Hexa-core	Y	Y	N	N/A	N/A
Prism setting	Y	Y	Y	N/A	N
Surface Meshing Parameters					
Ignore Size	N	N/A	N/A	N/A	N
Surface mesh type	N	Y	Y	Y	N/A
Prism Meshing Parameters					
Growth law	N/A	N/A	N/A	N/A	N/A
Initial height	N/A	N/A	N/A	N/A	N/A
Height ratio	N/A	N/A	N/A	N/A	N/A
Number of layers	N/A	N/A	N/A	N/A	N/A
Surface Parameters					
Max size*	Y	N/A	N/A	N/A	Y
Height	N/A	N/A	N/A	N/A	N
Height ratio	N/A	N/A	N/A	N/A	N
No. of layers	N/A	N/A	N/A	N/A	N
Tetra width	Y	N	N	N	N/A
Tetra size ratio	Y	N	N	N	N/A
Min Size Limit*	Y	N/A	N/A	N/A	Y
Max deviation*	Y	N/A	N/A	N/A	N
Surface mesh type	N	Y	Y	Y	N/A
Curve Parameters					
Max size*	Y	N/A	N/A	N/A	N
No. of nodes	N	N/A	N/A	N/A	N
Height	N/A	N/A	N/A	N/A	N

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Volume Meshing Methods				
Mesh Parameters	**Octree**	**Delaunay**	**Adv. Front**	**Hexa Dom.**	**Cartesian**
Height ratio	N/A	N/A	N/A	N/A	N
No. of layers	N/A	N/A	N/A	N/A	N
Tetra width	Y	N	N	N	N/A
Min Size Limit*	Y	N/A	N/A	N/A	N
Max deviation*	Y	N/A	N/A	N/A	N
Bunching	N	N/A	N/A	N/A	N
Densities*	Y	Y	N	N	Y
Connectors	Y	Y	Y	Y	N
**Scale Factor is applied*					

Table 7: Parameters for Other Meshing Methods

	Other Methods	
Mesh Parameters	**Prism**	**Blocking**
Global Parameters		
Max Element*	Y	Y
Min Size Limit*	N/A	N
Elements in gap	N	N
Refinement	N/A	N
Periodicity	Y	Y
Part Meshing Parameters		
Internal Wall	N/A	N/A
Thin Wall	N/A	N/A
Hexa-core	N/A	N/A
Prism setting	N/A	Y
Surface Meshing Parameters		

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Other Methods	
Mesh Parameters	**Prism**	**Blocking**
Ignore Size	N/A	N
Surface mesh type	N/A	N
Prism Meshing Parameters		
Growth law	Y	N
Initial height	Y	Y
Height ratio	Y	Y
Number of layers	Y	Y
Surface Parameters		
Max size*	N/A	Y
Height	Y	Y
Height ratio	Y	Y
No. of layers	Y	Y
Tetra width	N/A	N/A
Tetra size ratio	N/A	N/A
Min Size Limit*	N/A	N
Max deviation*	N/A	N
Surface mesh type	N/A	N
Curve Parameters		
Max size*	N/A	Y
No. of nodes	N/A	Y
Height	Y	Y
Height ratio	Y	Y
No. of layers	Y	Y
Tetra width	N/A	N/A
Min Size Limit*	N/A	N
Max deviation*	N/A	N
Bunching	N/A	N

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Other Methods	
Mesh Parameters	**Prism**	**Blocking**
Densities*	N/A	N
Connectors	N/A	N
**Scale Factor is applied*		

Global Mesh Size

The **Global Mesh Size** parameters affect meshers at the surface, volume, and inflation (prism) layer levels.

Global Element Scale Factor
multiplies other mesh parameters to globally scale the model. For a list of parameters affected by this scale factor, see the tables in Global Mesh Setup.

For example, if the **Max Element Size** of a given entity is 4 units, and the **Global Element Scale Factor** is 3.5, then the actual maximum element size used for meshing of that entity will be 4 * 3.5 = 14.0 units. The **Global Element Scale Factor** can be any positive real number, and it allows you to globally control the mesh size instead of changing the mesh parameters for different entities.

Display
when enabled, a reference mesh element will be displayed that corresponds to the specified element size.

Global Element Seed Size

Max Element
controls the size of the largest element. The largest element size in the model will not exceed the **Max Element** size multiplied by the **Global Element Scale Factor**. It is recommended that the **Max Element** value is a power of 2. Even if you specify a value other than a power of two, some meshers (Octree/Patch Independent) approximate the Max Element size to the nearest power of two while meshing.

Note

If the **Max Element** size is set to 0, the **Automatic Sizing** feature will be implemented. Autosizing temporarily sets a Global Max Element size, which produces a uniform mesh, if no surface or curve sizes are smaller. If most surface sizes are not set (< 22%), the autosizing will set the Global Max Element size to 0.025 * the bounding box diagonal of the geometry. If most surface sizes are set (> 22%), the autosizing will set the Global Max Element size to be the largest surface size that is specified.

If the Global Max Element size is too large (>= 0.1* the bounding box diagonal), and either no surface sizes are set or are also greater than this value, a message will appear informing you that the mesh size may be inadequate to represent the geometry and asking if you want to run with autosizing instead.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curvature/Proximity Based Refinement

when enabled, the mesh is automatically refined based on geometry curvature and proximity. This will result in larger elements on flat planar surfaces and smaller elements in areas of high curvature or within small gaps. The algorithm attempts to satisfy the **Refinement** and **Elements in Gap** settings, but is limited by the **Min size limit**. The effective Min size limit is scaled by the **Global Element Scale Factor** along with the **Max Element Size**. Entity Min size limits may be smaller than the global setting and will cause further local refinement.

Note

This option currently applies only to Octree and Patch Independent meshing.

All other mesh sizes will be rounded to the nearest power of 2 of the Min Size Limit value.

The following options are applicable when **Curvature/Proximity Based Refinement** is enabled:

Min size limit

specifies the size limit for the smallest element. Mesh elements will be limited from being subdivided smaller than this value. To see a mesh element of this size displayed on the screen, enable the **Display** option and click **Apply**.

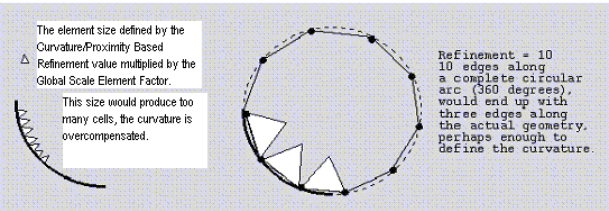
Elements in Gap

is used to force the Octree/Patch Independent mesher to create a defined number of elements in a gap (proximity based refinement). The specified value may not be possible if the Min size limit is too large, as the mesh can not be refined smaller than the Min size limit. Any positive integer can be entered for this option.

Refinement

defines the number of edges that would fit along a radius of curvature if that radius were extended out to 360 degrees. This is generally used to avoid having too many elements along a given curve or surface, if the **Min size limit** is too small for that particular curve. Any positive integer can be entered for this option. See the example in Figure 242: Refinement (p. 268).

Figure 242: Refinement


The element size defined by the Curvature/Proximity Based Refinement value multiplied by the Global Scale Element Factor.
This size would produce too many cells, the curvature is overcompensated.
Refinement = 10
10 edges along a complete circular arc (360 degrees), would end up with three edges along the actual geometry, perhaps enough to define the curvature.

Ignore Wall Thickness

prevents the Curvature / Proximity Based sizing function from refining for closely spaced parallel surfaces, though it will still resolve other forms of proximity. The **Elements in Gap** function may cause the model to over-refine within or around thin walled sections of a model. In these cases, the refinement results in relatively uniform elements and high mesh density in those areas. This can dramatically increase the element count. Enabling the **Ignore Wall Thickness** option will result in a mesh that uses larger elements in the thin walled area. There will still be refinement near the edges of the thin wall, but it will not refine most of the area. These larger elements will not be uniform and are likely to have lower quality. These higher aspect ratio tetra elements are also more likely to result in holes or non-manifold vertices in the thin wall, which can be addressed using the **Define Thin Cuts** option for Octree Tetra meshing.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the following examples, all refinement was controlled automatically by the **Curvature / Proximity Based Refinement** sizing function.

Figure 243: FEA Model with Ignore Wall Thickness Option Disabled

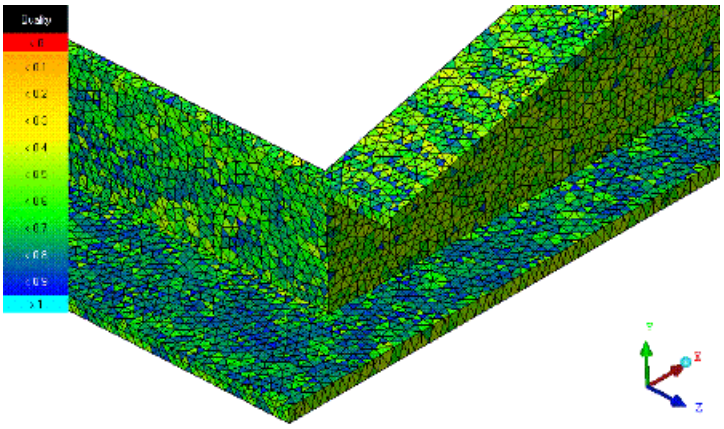
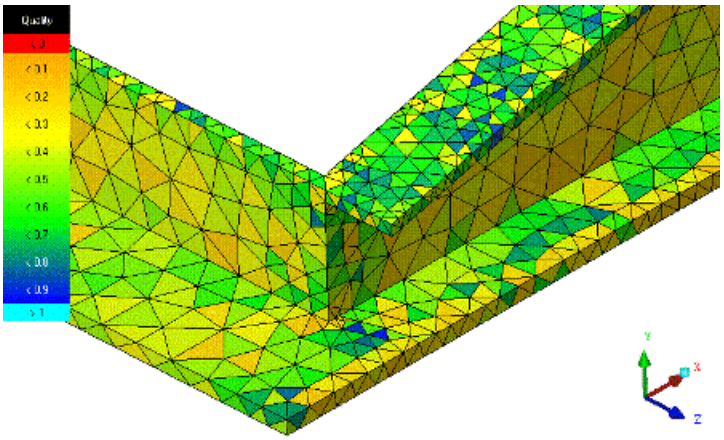
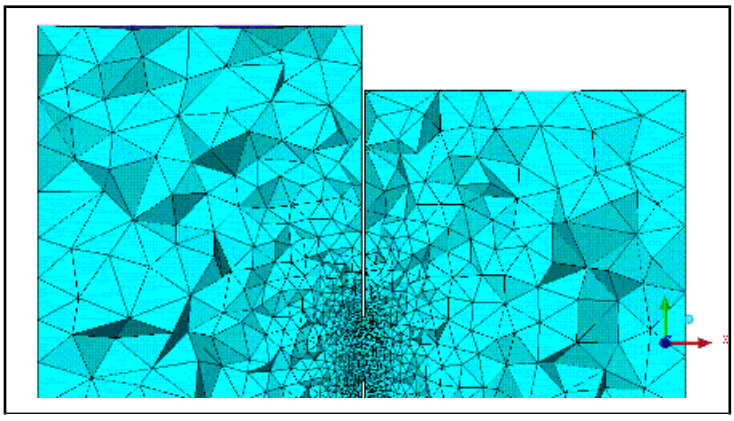


Figure 244: FEA Model with Ignore Wall Thickness Option Enabled

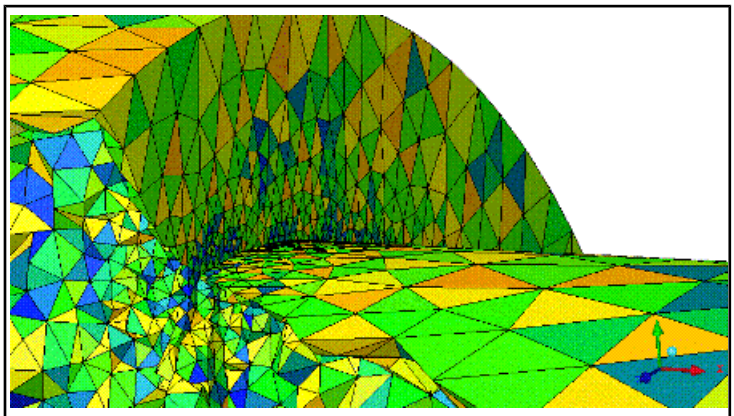


In the CFD example shown, there is a thin wall separating larger volumes. There is no need to refine the mesh for the wall thickness because meshing is not required within the wall. In this example, the quality is better when using **Ignore Wall Thickness** because of fewer size transitions.

Figure 245: CFD Model with Ignore Wall Thickness Option Disabled

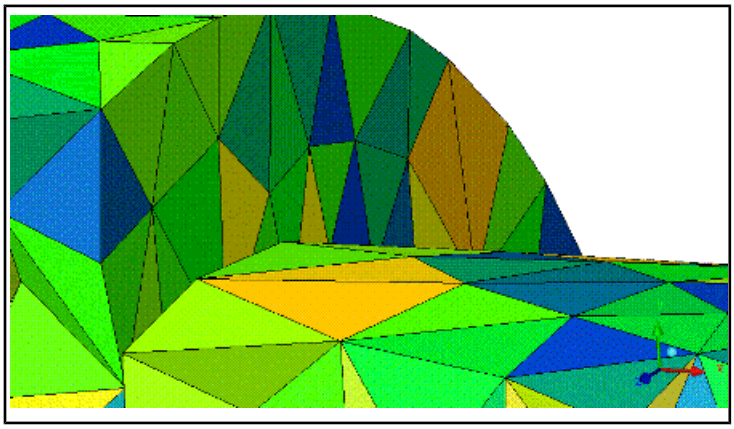


ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Another view of the same CFD model

Figure 246: CFD Model with Ignore Wall Thickness Enabled



Shell Meshing Parameters

The **Shell Meshing Parameters** control the various surface meshing algorithms. These parameters also control the surface mesh that is used for various volume meshers.

Mesh type

specifies the type of surface mesh. If an individual surface is assigned a different type of surface mesh, then the local control will override the global setting.

All Tri

meshes the geometry with a pure triangular mesh.

Quad w/one Tri

allows for a Quad Dominant mesh with one tri element per surface. The tri element allows for better transition between uneven mesh distribution on the loop edges. Automatic tri-reduction steps may even remove the tri element.

Quad Dominant

creates a quad mesh that allows for several transitional triangles. This mesh type is very useful in meshing complicated surfaces where a pure quad mesh may have poor quality.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All Quad
meshes the geometry with a pure quadrilateral surface mesh.

> **Note**
>
> This mesh type requires uniform sizes otherwise it will create some tri elements to maintain connectivity.

Mesh method
specifies the meshing method or algorithm to be used. If an individual surface is assigned a different type of mesh method, then the local control will override the global setting.

Autoblock
This method uses the mapped or block-based meshing algorithm. It automatically determines the best fit to obtain the defined minimum edge and orthogonality. For surface patches that cannot be mapped (having more or less than 4 corners), the Patch Dependent method is called through this block-based algorithm.

Patch Dependent
This is a free surface mesh generator that meshes closed regions called loops. Loops are created from surfaces or curves, and holes and internal curves are taken into account. The mesh is seeded according to the node spacing defined on the curves. Mesh from adjacent loops sharing a single curve is automatically joined together. This meshing method gives the best quad dominant quality while capturing surface details.

> **Note**
>
> For Patch Dependent meshing using Mesh Type of All Tri, the Curve Mesh Setup parameters of **Height**, **Height Ratio**, and **Num Layers** are respected, but quad elements will be used for the offset layers.
>
> If only **Height Ratio** is used in conjunction with the Global Shell Meshing Parameter **Adapt Mesh interior**, it can control the rate of transition for the All Tri or Quad Dominant mesh types.

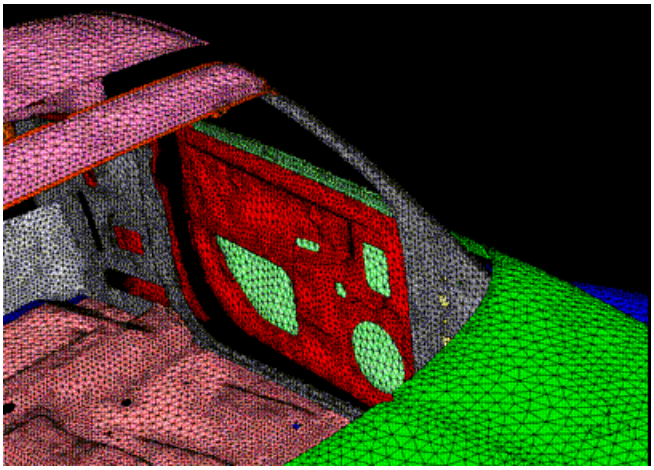
Patch Independent
Patch independent meshing is best for low quality geometry or surfaces with poor connectivity. It uses the Octree method to create a robust patch independent surface mesh. See The Octree Mesh Method in the ANSYS ICEM CFD User's Manual.

The geometry does not need to be a closed volume. If the **Mesh Type** is set to **Quad** mesh, the Patch Independent mesher will first generate a tri mesh, which can be automatically converted to a pure quad or quad dominant mesh.

> **Note**
>
> Patch independent meshing does not respect surface meshing parameters that have been set. However, it can take advantage of Octree parameters such as **Curvature/Proximity Based Refinement**, and Density Regions. Refer to the Table of Meshing Parameters.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 247: Example of Patch Independent Meshing



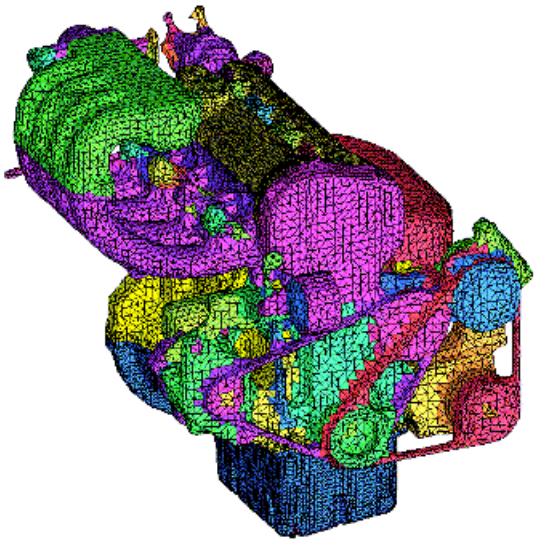
Shrinkwrap

Shrinkwrap meshing is best for geometry with gaps in STL representation. It uses the Cartesian method to initially generate all Quad mesh, with Quad dominant / All Tri options for better capturing of features. This generates watertight shell mesh with feature suppression. If the capture of greater detail is desired, then the Patch Independent Octree Tetra mesh would produce a better result, given that the model is of sufficient quality.

Note

The shrinkwrap mesher requires a closed volume input.

Figure 248: Example of Shrinkwrap Meshing



Autoblock Options

The **Autoblock** mesh method creates a 2D surface blocking in the background to generate 2D shell mesh. Since blocking is used in the background, you will essentially define the controls for the intermediate blocking file that will be generated. For more information on blocking and mesh generation, see Blocking > Create Block > 2D Surface Blocking.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Parameters

Ignore size
ignores sliver surfaces smaller than the specified value by merging the smaller loop with the adjacent loop.

Surface Blocking Options
The meshing algorithm on a surface to surface basis can be varied among the following options:

Free
All surfaces will be meshed similar to Patch Dependent Meshing.

Some mapped
Some surfaces will be meshed as orthogonally meshed surfaces, and the remainder will be meshed similar to Patch Dependent Meshing. Surfaces with primarily 4 corners and 4 sides are likely to be mapped (orthogonally meshed), even if they also have other shallow corners or multiple edges. Surfaces with more or less than 4 clear corners will be meshed as free blocks.

Mostly mapped
Most surfaces will be meshed as orthogonally meshed surfaces, and the remainder will be meshed similar to Patch Dependent Meshing. This is achieved by blocking subsurfaces, for example, a triangular surface may be blocked in a 3 block Y pattern (quarter O-grid), and a half circle may be blocked with a C-grid (half O-grid). In other cases, a surface with 5 or 6 corners may be split to create 2 mapped regions.

Merge mapped blocks
attempts to group mapped surfaces to form larger mesh regions.

Note

Blocks can be converted between free and mapped blocks using Edit Block > Convert Block Type.

Note

For more descriptions of the different block types, see User's Manual > Hexa > Hexa Block Types.

Patch Dependent Options

General Parameters

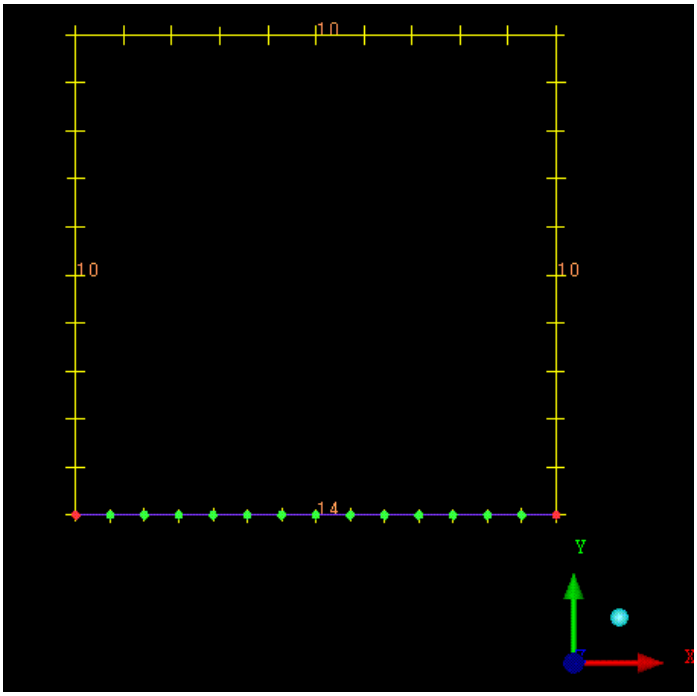
Ignore size
ignores sliver surfaces with an edge that is smaller than the defined value by merging the smaller loop with the adjacent loop.

Respect line elements
forces the surface mesher to respect nodes from given line elements on curves instead of creating new nodes. This is useful to connect a mesh to an existing mesh.

In Figure 249: Line Element Generated (p. 274), a curve of element count 14 was meshed and a line element was generated.

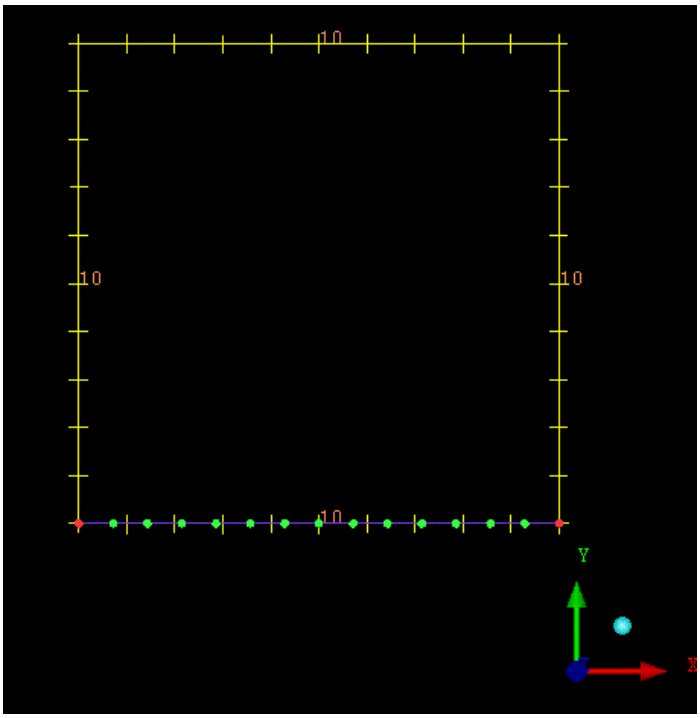
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 249: Line Element Generated



In Figure 250: Line Element Count Changed (p. 274), the element count was changed from 14 to 10.

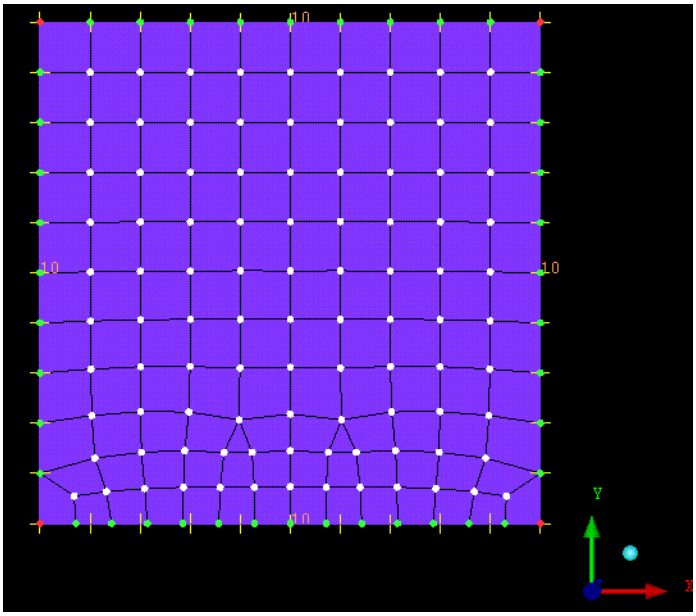
Figure 250: Line Element Count Changed



If the four curves are chosen for **Patch based surface meshing**, and **Respect line elements** is enabled, line element mesh will remain intact. The resulting mesh is shown in Figure 251: Respect Line Elements Enabled (p. 275).

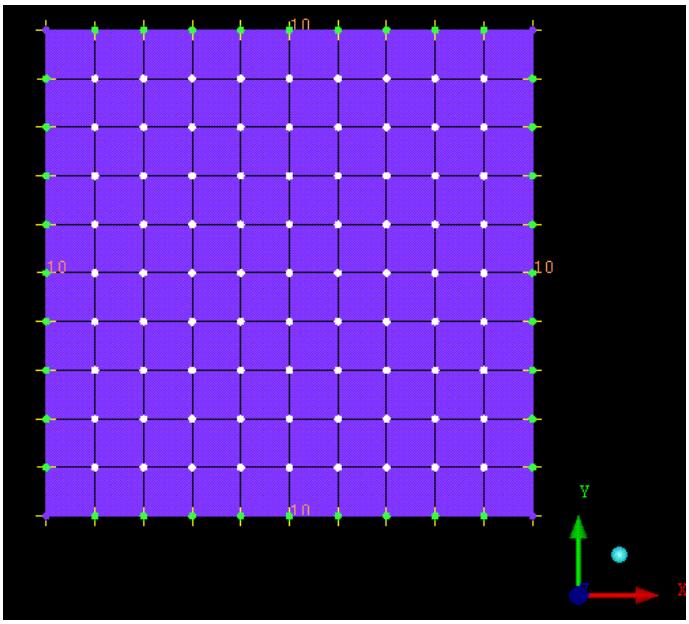
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 251: Respect Line Elements Enabled



When **Respect line elements** is disabled, the surface mesh created has ignored the line element mesh and readjusted the mesh on the curves based on the element count, as shown in Figure 252: Respect Line Elements Disabled (p. 275).

Figure 252: Respect Line Elements Disabled

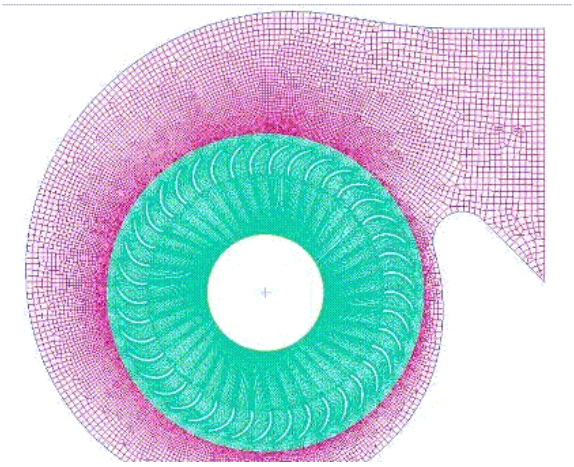


Respect line elements is useful to make sure that new mesh matches and connects with an existing mesh. This is particularly useful if the existing mesh came from another mesher, such as ANSYS ICEM CFD Hexa mesher, or from an outside product. This option is also helpful if there is a complex edge distribution that would be hard to match with the curve parameter controls.

In the example in Figure 253: Example of Respect Line Elements Option (p. 276), the inner fan was meshed using the ANSYS ICEM CFD Hexa mesher and the outer shroud was meshed with **Respect line elements** to create a node for node conformal mesh.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 253: Example of Respect Line Elements Option



Quadratic elements

when enabled, generates the patch dependent surface mesh with mid side nodes (quadratic elements), such that triangles have 6 nodes (3 corners and 3 mid side nodes) and quads have 8 nodes (4 corners and 4 mid side nodes). The mid side nodes are projected to the surface and the resulting edge shape is quadratic (rather than linear). When this option is disabled, the patch dependent surface mesh consists of linear triangles and quads, each with 3 or 4 corner nodes, respectively.

Note

Typically FEA solvers prefer quadratic elements. Most CFD solvers do not support quadratic elements at all.

Tip

You can also use the Create Mid Side Nodes (p. 546) option to convert linear elements to quadratic.

Boundary Parameters

Protect given line elements

is available when **Ignore size** is set and **Respect line elements** is enabled in the **General** section of the **Shell Meshing Parameters** list. If enabled, it will protect existing line elements that are smaller than the **Ignore size** value from being removed.

Smooth boundaries

smooths the surface mesh boundaries after meshing. This will typically give better quality mesh, but it may not respect the initial node spacing.

Allow free bunching

if enabled, it allows free bunching for patch independent (Octree, with or without Tetra) surfaces. If disabled, curve bunching is done by the Patch Dependent mesher and respected by Patch Independent functions.

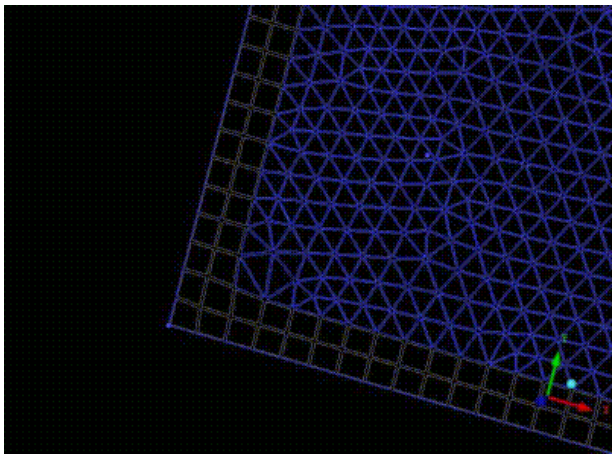
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Offset type

Standard

Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front may not be identical as the number of nodes on the initial boundary.

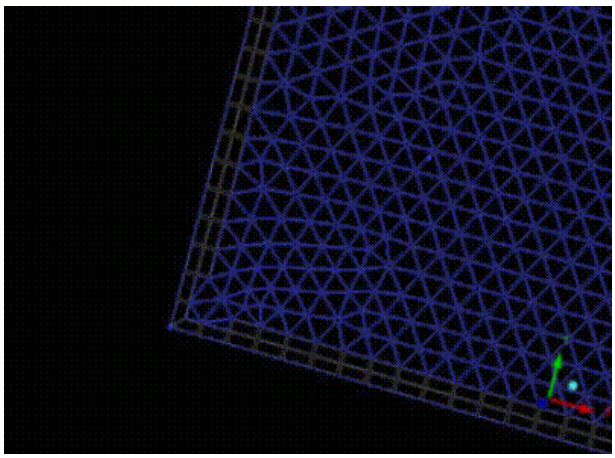
Figure 254: Standard Offset Example



Simple

Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front is identical to the number of nodes on the initial boundary.

Figure 255: Simple Offset Example

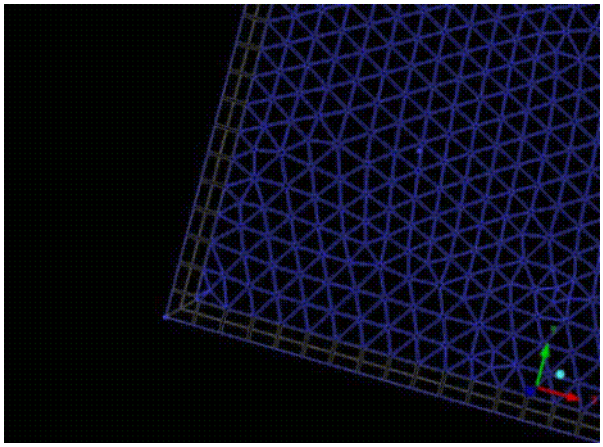


Forced Simple

This option is the same as simple offset, but without collision checking.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 256: Forced Simple Offset Example



Interior Parameters

Force Mapping

If the boundary is nearly quadrilateral, the mesher forces the generation of structured mesh instead of unstructured mesh up to this specified block quality. Default value is 0. For hybrid meshes a value of 0.2 is preferred.

Max nodes adjustment

For opposite boundaries with differing node counts, this function will calculate the ratio of the node counts as a percentage. For all ratio percentage values less than this specified value, mapping will be applied to the mesh. For all ratio percentage values greater than this specified value, mapping will not be applied.

Project to surfaces

If enabled, it will allow the mesher to project the mesh to the surface.

Note

Disable this if the geometry does not have surfaces.

Adapt mesh interior

Uses the surface sizes to coarsen the mesh internally. For example, if curve size is set to 1, and surface size to 10, then the mesh will start with a mesh size of 1 on the curves, but transition to 10 in the middle of the surface. This is more effective on larger surfaces where the element reduction is more dramatic.

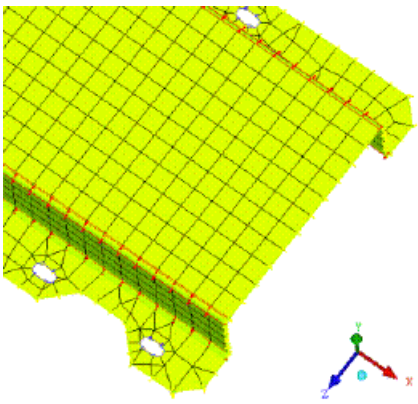
The default growth rate for the transition to the surface max size is 1.5. This growth rate can be adjusted by setting the surface **Height Ratio** to between 1.0 and 3. Sizes below 1.0 are inverted (i.e., 0.667 = 1.5). Sizes above 3 are ignored and the default is used. If this option is enabled, **Force Mapping** is disabled on surfaces whose maximum size setting exceeds its perimeter curve sizes by a factor of 2 or more.

The difference in the transition rate is shown in Figure 257: Example of Adapt Mesh Interior option (p. 278):

Figure 257: Example of Adapt Mesh Interior option

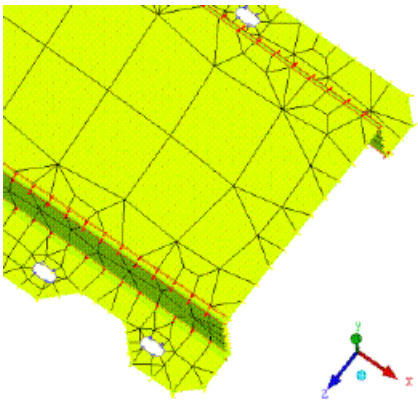
Adapt Mesh Interior option disabled

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

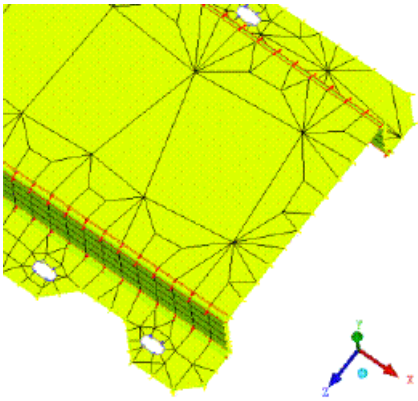


Adapt Mesh Interior option enabled

Default Surface Height Ratio = 1.5



Surface Height Ratio = 3



Note

The **Max Element** size specified in the General Parameters takes precedence over this value.

Note

For All Quad, Quad Dominant, and Quad with one Tri mesh, the **Force Mapping** option has priority over this option. To apply the **Adapt mesh interior** option to these types of mesh, set the **Force Mapping** value to 0.

Orient to surface normals

orients the shell normals in the same direction as the surface normals. This option is enabled by default.

Repair Parameters

Try harder

- **Level 0**

  If the mesher fails, no further meshing step is tried. The problem(s) will be reported.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Level 1**

  If the mesher fails, simple triangulation is attempted to fix the problems (for “All Tri” and “Quad Dominant” mesh types only).

- **Level 2**

  All Level 1 steps will be completed. If necessary, all Level 1 steps will be retried without merging at dormant curves.

- **Level 3**

  All Level 2 steps will be completed. If necessary, surface meshing will be retried with the Tetra mesher.

Improvement level

- **Level 0** does pure Laplacian smoothing. Nodes are moved while keeping the mesh topology unchanged.
- **Level 1** will mesh any failed loop with an STL method if element types **Quad Dominant** or **All Tri** has been selected. This makes the mesher more robust. Very bad quadrangles will be split into triangles.
- **Level 2** will operate as Level 1, and also will combine triangles to quadrangles and will split more bad quadrangles into triangles. This option is most used.
- **Level 3** will operate as Level 2, and also will move nodes off curves to improve the quality.

Respect dormant boundaries
If enabled, all dormant curves and points are included in the mesh boundary definitions. The default is off.

Relax dormant boundaries for smoothing
If **Mesh dormant** is enabled, then this option allows nodes on dormant curves and points to be moved in order to improve smoothing.

Patch Independent Options

This method uses the Octree Tetra process to create a robust patch independent surface mesh. See The Octree Mesh Method in the ANSYS ICEM CFD User's Manual.

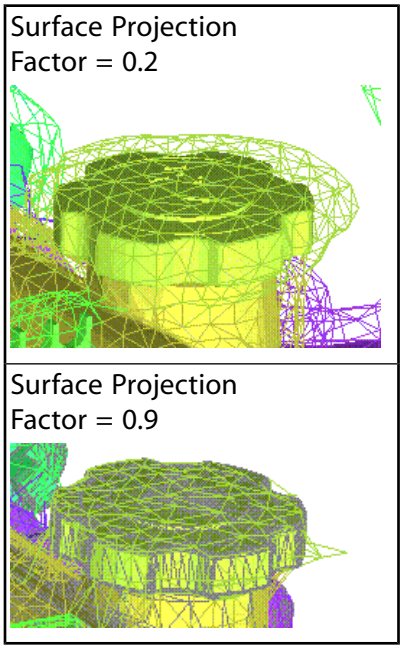
Shrinkwrap Options

Num. of smooth iterations
is the number of smoothing iterations that will be performed to improve mesh quality.

Surface projection factor
controls the tightness of the shrinkwrap to the geometry. Values can range from 0 to 1.0, where a value of 0 indicates shrinkwrap totally free from the geometry, and a value of 1.0 indicates shrinkwrap that is strictly on the geometry. The example in Figure 258: Surface Projection Factor (p. 281) illustrates the results of using different values for the **Surface projection factor**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 258: Surface Projection Factor



Volume Meshing Parameters

The **Volume Meshing Parameters** control the various volume meshing algorithms. Each mesh type has different options, as described in the following sections.

Tetra/Mixed
Hexa-Dominant
Cartesian

Tetra/Mixed

The following **Mesh Methods** are available for Tetra meshing:

Robust (Octree)
Quick (Delaunay)
Smooth (Advancing Front)

Robust (Octree)

The **Robust (Octree)** option will generate a tetra mesh using a top-down meshing approach. An Octree mesher does not require an existing surface mesh because one is created by the Octree process. It will accept a variety of parameters in a more general way. For instance, curve sizes are respected, but specific curve node spacing distributions are not. For a better understanding of the Octree meshing methodology, see The Octree Mesh Method in the ANSYS ICEM CFD User's Manual.

Run as batch process
allows you to run the meshing operation in batch mode.

Fast transition
enables a faster transition from the more refined elements to coarser elements instead of a more gradual transition when computing the mesh. This will reduce the number of elements in the overall mesh. This option is particularly useful if you plan to delete the Octree volume mesh anyway since it saves time and memory during mesh generation.

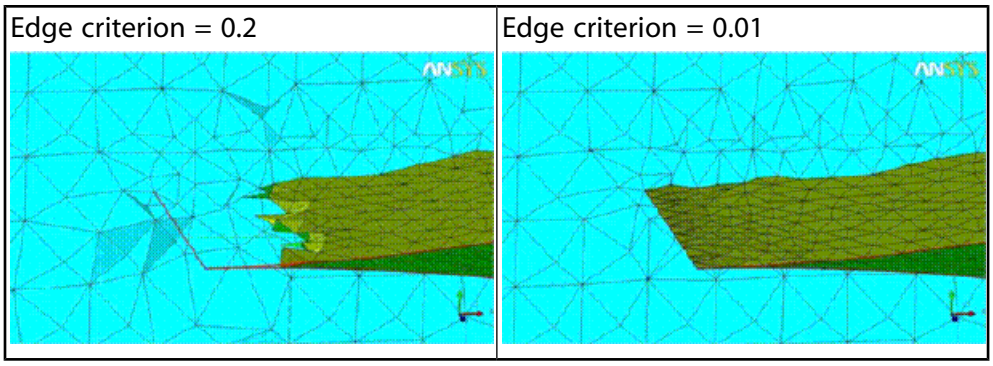
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge criterion

determines to what extent a tetra is cut to represent geometry. The entered value is a factor of the tetra edge. After subdivision, if a tetra edge intersects an entity (surface), the tetra will be cut if the subdivision of the edge from the intersection is less than the prescribed value.

The values can range from 0 to 1. A larger value (e.g., 0.5) may be prescribed if you want to ignore inexact duplicates, however geometry may not be properly captured if the value is too large. Increasing the value will increase the nodes moving to fit to the geometry. Reducing the value will increase refinement near entities, and will also reduce non-manifold vertices in trailing edges. The default of 0.2 is adequate for most cases. This value is useful in cases where thin cuts fail or are difficult to setup.

Figure 259: Edge Criterion



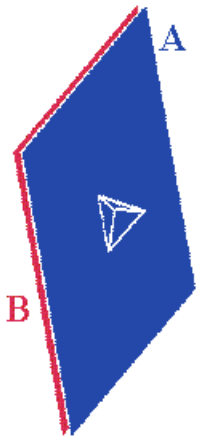
Define thin cuts

allows you to resolve gaps based on pairs of surface or curve parts which are close to each other. The thin cut definition prevents surface elements from spanning between nodes on the two parts specified. This helps prevent non-manifold nodes in areas where the mesh size may be larger than the gap.

Figure 260: Thin Cuts (p. 283) shows an example of thin cuts. If the face of a tetra element has a surface/line/node on part A, then it cannot have a surface/line/node in part B.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

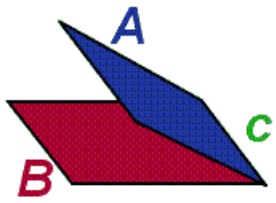
Figure 260: Thin Cuts



Note

If the surfaces of the two parts, A and B, meet (see Figure 261: Thin Cuts for Intersecting Parts (p. 283)), then the contact curve must be in a third part, C, or the thin cut will fail.

Figure 261: Thin Cuts for Intersecting Parts



Clicking the **Define thin cuts** button will bring up the **Thin cuts** window where pairs of parts can be selected or entered to define a thin cut "pair". A part can belong to more than one thin cut pair.

The following limitations apply to the definition of thin cuts:

- If two surfaces approach asymptotically, the thin cut definition may eventually fail.
- If the separation rule of thin cuts is violated at any point, all thin cuts will fail and be deactivated.

Due to the existing limitations with defining thin cuts, in some cases, it may be more effective to adjust the edge criterion.

Edit entry
allows you to enter or edit the names of the parts in a pair.

Select
allows you to select parts from a window displaying all the part names. The pair of parts selected should not be touching. If two surface parts meet along an edge (such as a sharp cusp) the curves and points between the parts must be in a third part name.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add
adds the thin cut pair selected or entered.

Modify
allows you to modify a thin cut pair.

Delete
allows you to delete a thin cut pair definition.

Done
saves the thin cut pair definitions and closes the window.

Cancel
cancels the thin cut pair definitions.

Smooth mesh
when enabled, the mesh is smoothed after finishing subdivision down to the specified mesh sizes.

Iterations
number of iterations to be performed for the smoothing to match the supplied **Min quality**.

Min Quality
all elements with quality less than the specified value will be smoothed. The default is 0.4.

Coarsen mesh
is a useful option if the geometry to be captured is difficult to mesh with the desired coarse mesh size. This allows the Octree process to fit to the geometry and flood fill with a finer mesh and then automatically coarsen it. For instance, it could mesh to capture features with approximately 150k elements and then automatically coarsen down to 15k elements while still maintaining the features. This may be needed in some situations, but it is recommended that you mesh without this option and coarsen the mesh with the **Edit Mesh** tools where there are more controls available.

Iterations
number of iterations to be performed for the coarsening to meet the specified **Worst Aspect Ratio**.

Worst Aspect Ratio
a limiting factor for mesh coarsening (any elements below this quality will be excluded from the next iteration of coarsening).

Fix Non-manifold
if enabled, the mesher will do additional work to try to fix non-manifold elements. These are only a possible problem, depending on your model and solver. See here for a description of non-manifold elements. It is strongly recommended to leave this option on and that you check for non-manifold elements after the mesh is computed.

Note

You may still have non-manifold elements for several reasons. They may be representative of the true geometry, and acceptable to the solver and therefore not a problem. It is also possible that the meshing parameters may be too coarse to cope with the geometry; in which case you could reduce the mesh size and/or apply thin cuts and/or reduce the edge criterion to allow for subdivision within the thin space. If only a few non-manifold

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements exist, you can repair them with mesh editing. Try splitting nodes or delete the non-manifold elements and remesh locally.

Close Gaps

is used to close gaps in the surface mesh between different materials. If enabled, triangles will be created to fill these gaps.

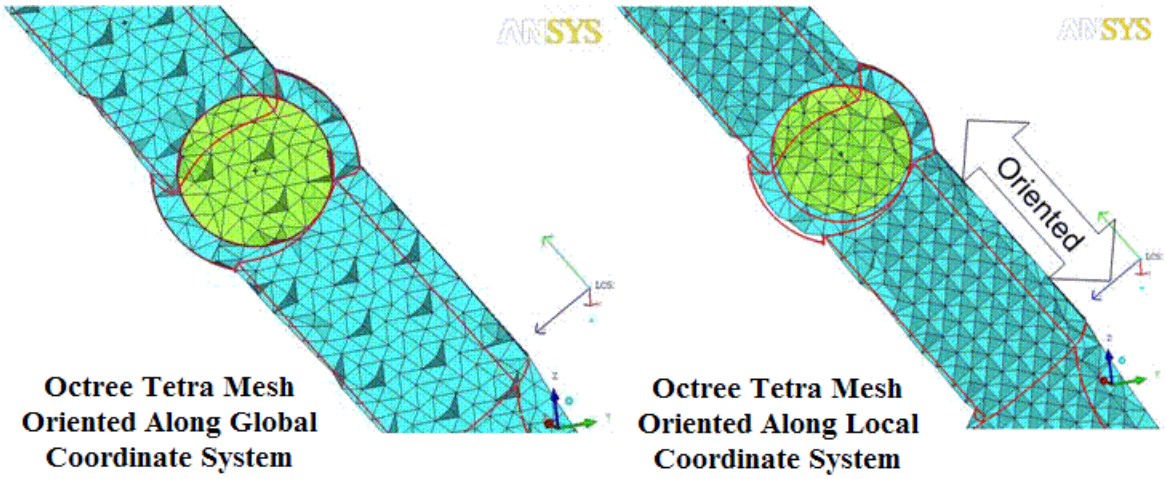
Fix Holes

if disabled, the mesher will not perform extra steps to try to close holes to form a watertight volume. If enabled, the mesher will perform extra steps to close holes with a clear solution. If the mesher detects leakage with an unclear solution, it will interactively prompt the user for the solution.

Use active local coordinate system

allows you to orient the Octree mesh along the active local coordinate system instead of along the global coordinate system. If the LCS is not Cartesian, the equivalent Cartesian LCS is used.

Figure 262: Orienting Octree Tetra Mesh Along LCS

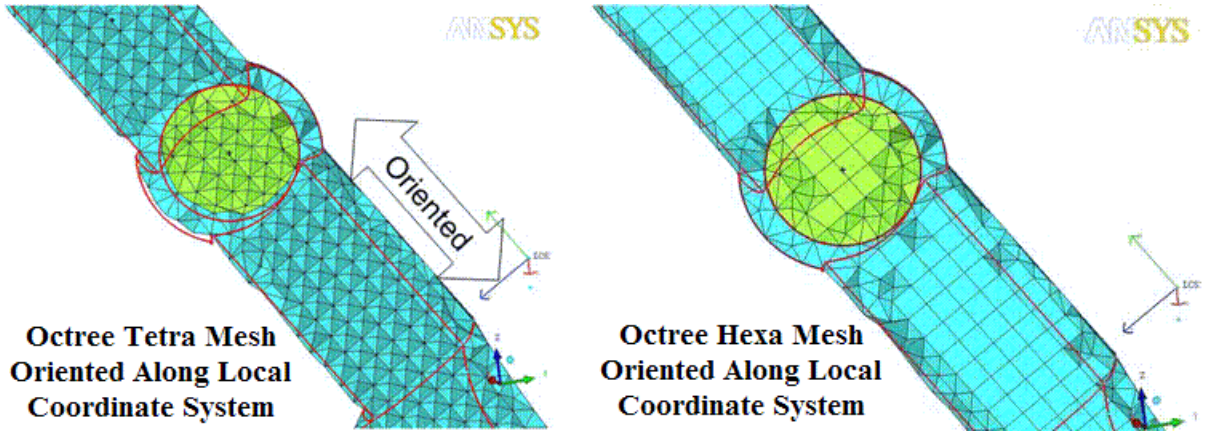

Oriented
Octree Tetra Mesh Oriented Along Global Coordinate System
Octree Tetra Mesh Oriented Along Local Coordinate System

Note

This option is helpful when converting oriented Octree tetra mesh to oriented hexa hybrid mesh (**Convert Tetra to Hexa** method – 12 tetra to 1 hexa).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 263: LCS Oriented Octree Tetra Mesh Converted to LCS Oriented Hexa Mesh



In such cases, make sure the **Use active coordinate system** option is enabled for the **12 tetra to 1 hexa** method in the **Convert Tetra to Hexa** DEZ (see the Convert Tetra to Hexa section).

Quick (Delaunay)

The **Quick (Delaunay)** option will generate a tetra mesh using a bottom-up meshing approach (Delaunay Tetra algorithm). This algorithm requires an existing, closed surface mesh. If this has not yet been created, it will automatically create the surface mesh from the geometry as defined by the Global Mesh Setup settings (or the Surface Mesh option). The volume mesh will then be generated from this surface mesh. You can also run this in two steps by creating/importing a surface mesh first and then running this mesher. If a surface mesh exists, you can also specify the **Input** as **Existing Mesh** when **Compute Mesh** is applied.

Note

A closed surface mesh is needed to contain the volume mesh. You can run a mesh check for single edges, overlapping elements or duplicate elements (**Edit Mesh > Check Mesh**). Single edges may appear on the edges of internal baffles, but should not be found on the outer perimeter of the model or the Delaunay fill will fail.

Delaunay Scheme
allows you to select the Delaunay scheme to be used.

Standard
uses the standard Delaunay scheme with a skewness-based refinement.

TGlib
uses the latest TGrid Delaunay volume grid generation algorithm that utilizes a more gradual transition rate near the surface, and a faster transition rate towards the interior. Like the standard Delaunay scheme, it uses a skewness-based refinement.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use AF
uses the latest TGrid Advancing Front Delaunay algorithm which has smoother transitions than the pure Delaunay algorithm.

Note

The **TGlib** options will be ignored when the **Create Hexa-Core** option is enabled for the **Quick (Delaunay)** method.

Memory scaling factor
The initial memory requirements will be multiplied by this factor. If no value is supplied, the default is taken as 1. The initial memory requirement is calculated from the surface mesh (or from volume mesh if it is supplied). If the initial memory calculation falls short, it will double it and try again. It will restart 3 times before failing due to insufficient memory allocation.

Spacing Scaling Factor
The rate at which the tetra grow from the surface mesh. It is similar, but not mathematically identical, to the expansion factor used by other algorithms. This value directly affects the number of tetra elements generated.

Fill holes in volume mesh
This is for use on an existing tetra mesh with internal voids (cavity re-meshing). This will fill the voids without regenerating the full tetra domain. You could delete the tetra mesh in a region, insert a new surface mesh component and then cavity re-mesh to the existing tetra mesh. Alternatively, this could be used simply to remesh a region of poor tetra mesh.

Mesh internal domains
If this option is enabled, the Delaunay tetra mesher will also attempt to fill internal volume regions. With this option disabled, only those volumes adjacent to external shell elements will be meshed.

Flood fill after completion
This option only pertains to models with multiple material points. If this option is enabled, the volume mesh will be assigned to different volume parts based on material point containment.

Verbose output
When enabled, the mesher outputs more detailed messages to help in debugging any potential problem. In general, you should not need to have this option enabled, but it may help to enable this for debugging.

Smooth mesh
When enabled, the mesh is smoothed after finishing subdivision down to the specified mesh sizes.

Iterations
Specifies the number of iterations to be performed for the smoothing to match the supplied **Min quality**.

Min Quality
All elements with quality less than the specified value will be smoothed. The default is 0.4.

Smooth (Advancing Front)

The **Smooth (Advancing Front)** option will generate a tetra mesh using a bottom-up meshing approach using the Advancing Front Tetra mesher. The surface mesh will be created as defined by the Global Mesh Setup settings (or Surface Mesh option). The volume mesh will then be generated from this surface

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh. If a surface mesh exists, you can also specify the **Input** as **Existing Mesh** when **Compute Mesh** is applied.

This meshing method results in a more gradual change in element size. The initial surface mesh should be of fairly high quality.

Note

The surface mesh should be one enclosed volume with no single edges, multiple edges, non-manifold vertices, overlapping elements or duplicate elements. Sudden changes in element size, either adjacent to one another or across a narrow volume gap, can cause quality issues or even failure.

The surface mesh must be either tri or quad elements for the Advancing Front mesh method.

Expansion Factor
The ratio at which to grow the tetra from the surface mesh. The default is 1. This value directly affects the number of tetra elements generated.

Do Proximity Checking
This option will check the proximity between nodes to prevent clustering or stretching so that small gaps are filled properly. Enabling this option will result in a longer meshing time.

Flood fill after completion
This option only pertains to models with multiple material points. If this option is enabled, the created mesh will be assigned to their different volume parts.

Verbose output
When enabled, this option writes more messages to help in debugging any potential problem. In general you should not need to have this option enabled, but if the mesher has a problem meshing, it may help to enable this for debugging.

Hexa-Dominant

The **Hexa-Dominant** option will generate a hexa-dominant mesh using a bottom-up meshing approach. The Hexa-Dominant mesher starts with surface quad dominant mesh and uses an Advancing Front scheme to fill as much of the volume as possible. For simple volumes, it can fill it completely. For more complicated volumes, it usually fills several layers in from the surface with hexa elements and then fills the middle with tetras and pyramids. Then a diagnostic is run, and if those central elements are poor, the inner volume will be meshed again with the Delaunay mesher. The surface mesh will be created with the parameters defined under Global Mesh Setup or Surface Mesh Setup. The volume mesh will then be generated from this surface mesh. If a surface mesh exists, you can also choose to specify **Existing Mesh** for the **Input**.

Remesh Center
deletes tetra elements from the center region of the mesh, and remeshes it using the Delaunay mesher. This will result in better quality mesh at the center region.

Cartesian

The **Cartesian** option will generate a Cartesian mesh using a top-down meshing approach. This creates pure Cartesian and unstructured Cartesian meshes with H-Grid refinement. The mesher works by continuously refining the initial grid in a binary fashion in each dimension, and eliminating the non-volume

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cells, up to the specified maximum refinement. The Cartesian mesher does not require an existing surface mesh and will ignore meshing parameters defined to control local surfaces.

The following mesh methods are available.
Body-Fitted
Hexa-Core

Body-Fitted

The **Body-Fitted** option creates unstructured hexa mesh of uniform size based on Cartesian mesh and fits it to the geometry. This works for both CAD and STL geometries. The mesher can handle "dirty" geometries as long as the cell size is larger than the gap size.

Note

The cell size should be smaller than the thickness of the model.

Projection Factor
controls the tightness of the body-fitted Cartesian mesh to the geometry. Values can range from 0 to 1.0, where a value of 0 indicates Cartesian mesh totally free from the geometry, and a value of 1.0 indicates mesh that is strictly on the geometry.

Split Degenerate
splits boundary quad and hexa elements with flat corners (triangular shaped), and propagates this split to produce higher quality elements. This option does not introduce pyramids or tetra elements.

Note

The shorter edge length introduced by this option may pose a problem for some users such as reducing the time step for explicit solvers. Also, the quality improvement is usually not as good as the improvement produced by the Create Pyramids option. It is recommended that you try different options to determine which produces the best result for a particular topology and for a particular solver/physics.

Figure 264: Body-Fitted Cartesian Mesh with Boundary Hexa Elements (p. 290) shows the BFC mesh having boundary hexa elements with flat corners.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 264: Body-Fitted Cartesian Mesh with Boundary Hexa Elements

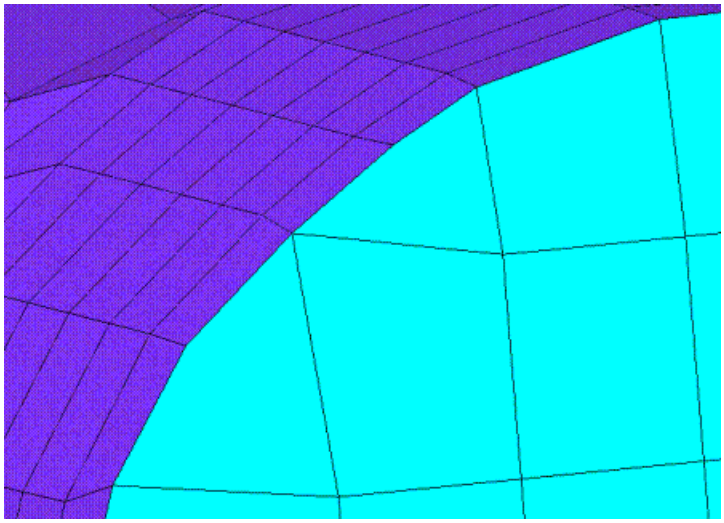
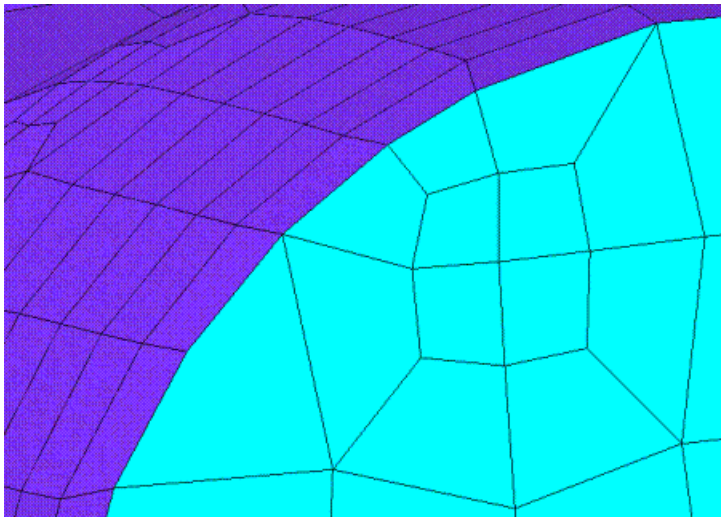


Figure 265: Split Degenerate Option (p. 290) shows the results of using the **Split Degenerate** option. The boundary hexa element has been split and the split is propagated through the neighboring elements.

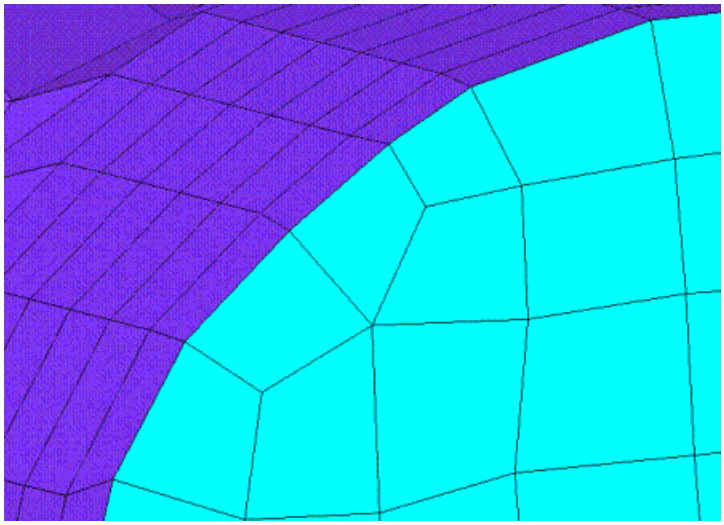
Figure 265: Split Degenerate Option



For this model, the use of an inflation layer proves to provide the best quality, as shown in Figure 266: BFC Mesh with Inflation Layer (p. 291).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 266: BFC Mesh with Inflation Layer



Create Pyramids

remeshes bad quality hexa elements (with determinant quality less than 0.05) using a Delaunay algorithm. This effectively replaces the poorest hexa elements with higher quality tetra and pyramid elements.

Refinement Type

Uniform

creates hexa elements of uniform size.

2-to-1

creates hexa elements of varying size with 2-to-1 size transition. This will introduce hanging nodes in the mesh.

3-to-1

creates hexa elements of varying size with 3-to-1 size transition. This mesh can be made conformal using the **Edit Mesh > Resolve refinements** option for limited configurations.

Aspect Ratio

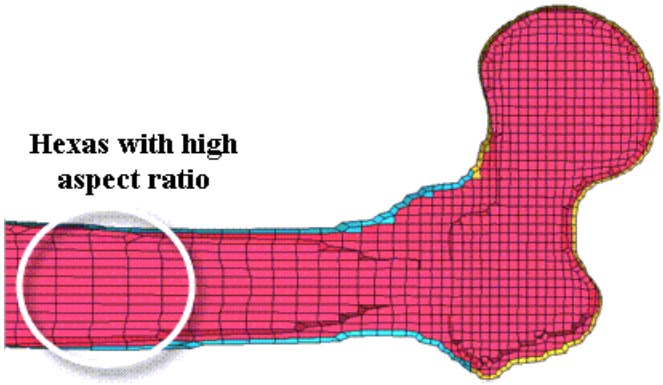
allows you to control the aspect ratio of the otherwise uniform Cartesian mesh. This can be used to fill the volume more efficiently in cases where there is a strong gradient aligned with the coordinate system. The default setting is **`1 1 1`**. Changing the aspect ratio allows you to stretch the Cartesian grid. For example, setting the aspect ratio to **`1 2 0.75`** will result in a Cartesian mesh that is twice as long in the Y direction as the X direction and 0.75 times the size in the Z direction. This is applied to the entire model.

You can also create a Cartesian file with varying aspect ratio. Use the Hexa blocking tools to create a block with splits and edge parameters to control the distribution. Then, use the **File > Blocking > Write Cartesian Grid** option to convert the blocking to a Cartesian grid file. You can then refer to this file when computing the grid (see the Cartesian file (p. 344) option for details).

Figure 267: Hexa Mesh With Varying Aspect Ratio (p. 292) shows the Cartesian mesh for a Femur, with the aspect ratio varying along the shaft.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 267: Hexa Mesh With Varying Aspect Ratio



Project Inflated Faces

allows full projection to the surface for inflated faces rather than limiting the projection with the projection factor.

Outer Bounding Box

allows you to specify the outer mesh region for external flow meshes by setting the min-max coordinates of the flow region. This can be used instead of explicitly creating the tunnel geometry.

min-max Coordinates

specify the explicit coordinates or click the selection icon to pick diagonal min and max coordinates to specify the outer bounding box size.

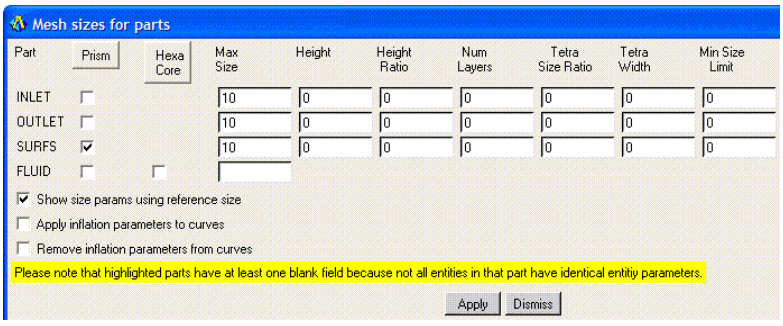
Use active local coordinate system

allows you to orient the mesh along the active local coordinate system instead of along the global coordinate system. LCS are translated to equivalent Cartesian coordinates.

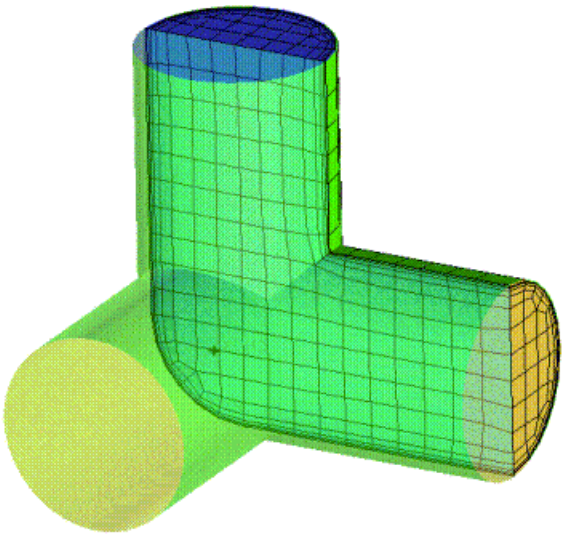
Boundary Layers

The Body-Fitted Cartesian mesher can create a layer of hexa elements parallel to the wall. These are necessary for properly capturing curvature and can be subdivided to get boundary layers. For some geometry parts, such as inlets, outlets and "flat" sides, you may not want to have this offset. However, you may want to apply inflation to flat parts if they are not in the Cartesian planes (X Y Z) to prevent stair-stepping. To determine if a part will have a single offset layer (with uniform aspect ratio), enable **prism** for the appropriate parts in the Part Mesh Setup dialog. For ANSYS ICEM CFD, standard prism controls do not apply for Body-Fitted Cartesian meshing. Instead, if the **prism** option is enabled for a specific part, and then meshed using the Cartesian Body-Fitted method, the boundary layer will be offset. The boundary layer is determined on a part by part basis, so parts must be created and surfaces assigned appropriately.

Figure 268: Example of Boundary Layers in Body-Fitted Cartesian Mesh



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In Figure 268: Example of Boundary Layers in Body-Fitted Cartesian Mesh (p. 292), the **prism** option is enabled for the SURFS part containing the walls of the corner pipe but disabled for the INLET and OUTLET parts. When the Body-Fitted Cartesian mesh is computed, the boundary layer is grown along the SURFS part to better capture curvature, but terminates at the INLET and OUTLET. This single boundary layer can then be split to increase near wall resolution if needed.

Define Thin Cuts

The **Define thin cuts** option allows you to prevent elements stretching across a gap. You can define the thin cut pair using the **Define thin cuts** button in the **Tetra/Mixed Meshing** parameters for the **Robust (Octree)** mesh method (see the Define thin cuts (p. 282) option).

Note

The thin cuts option works best for regions of the mesh which will be cut away, rather than thin solid regions where the mesh is to be retained.

Hexa-Core

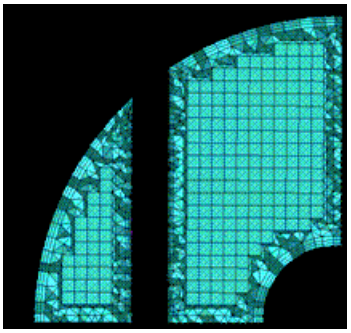
The **Hexa-Core** option will generate a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

Note

Hexa-Core is implemented as part of Tetra meshing. Under **Compute Mesh > Volume Mesh**, the **Create Hexa-Core** option can be selected when using Tetra/Mixed Mesh Type, and Robust (Octree) or Quick (Delaunay) Mesh Methods.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 269: Example of Hexa-Core Mesh



Fill holes in volume mesh
will remesh only the missing tetra portions. This option is required only for multiple tetra fill volume regions. It is disabled by default, as it is not necessary for single volume fill (internal meshes).

Refinement Type

Uniform
creates hexa elements of uniform size.

2-to-1
creates hexa elements of varying size with 2-to-1 size transition. This will introduce hanging nodes in the mesh.

3-to-1
creates hexa elements of varying size with 3-to-1 size transition. This mesh can be made conformal using the **Edit Mesh > Resolve refinements** option for limited configurations.

Outer Bounding Box
allows you to specify outer mesh region for external flow meshes.

min-max Coordinates
specify the explicit coordinates or click the selection icon to pick diagonal min and max coordinates to specify the outer bounding box size.

Use active local coordinate system
allows you to orient the mesh along the active local coordinate system instead of along the global coordinate system.

Prism Meshing Parameters

The **Prism Meshing Parameters** include global prism settings as well as more advanced options. You can create prisms from existing volume or surface mesh.

See the Prism Mesh section in the User's Guide for more general information on Prism Meshing.

Global Prism Settings

Growth Law
determines the height of the layers given the initial height and height ratio.

- **Linear**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The prism height of a particular layer is defined by h(1+(n-1)(r-1)), where h=initial height, r=height ratio, and n=layer number. The total height at layer n is:

nh((n-1)(r-1) + 2)/2.

- **Exponential**

  The prism height of a particular layer is defined by $h*r^{(n-1)}$, where h=initial height, r=height ratio, and n=layer number. The total height at layer n is:

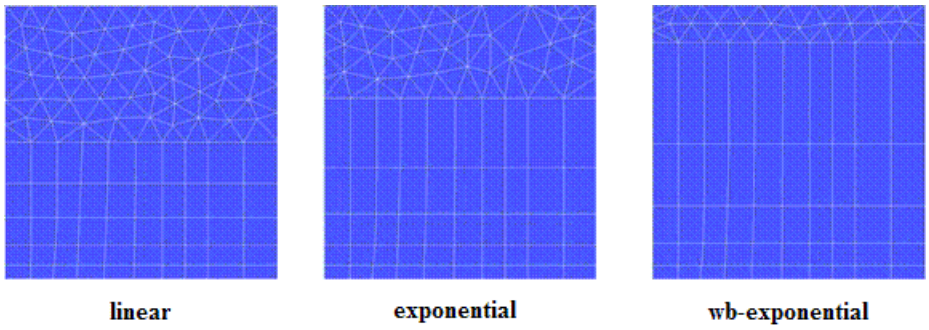
  $h(1-r^{n})/(1-r)$.

- **WB-Exponential**

  This is the exponential growth law as defined in ANSYS Workbench. The prism height of a particular layer is defined by h*exp((r-1)(n-1)), where h = initial height, r = height ratio, and n = layer number.

Figure 270: Prism Growth Law (p. 295) shows a box with edge length = 1, h = 0.05, r = 1.5, and n = 5, with different growth laws applied.

Figure 270: Prism Growth Law



Initial height
is the height of the first layer of elements.

Note

If the initial height is set to 0, it will be automatically determined by the local triangle size, where the height of the last layer will be the minimum attached triangle edge length times the **Prism height limit factor**.

Height ratio
is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the height of the next layer.

Number of layers
is the number of layers to be grown from the surface or curve.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total height
is the total height of all the prism layers.

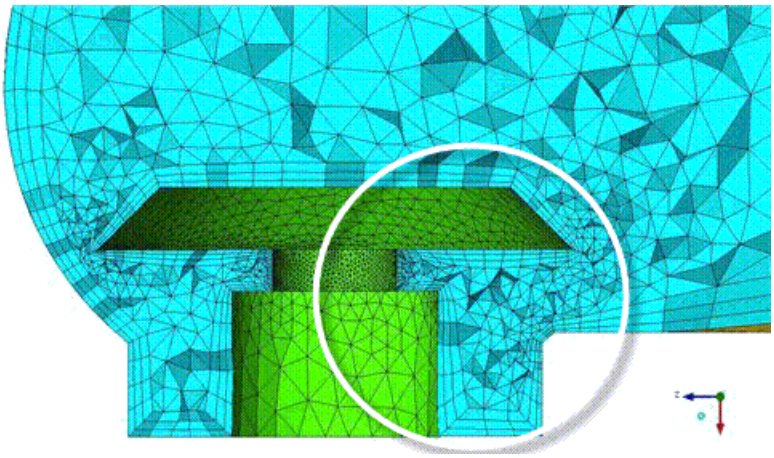
Note

If neither the initial height nor the total height is set, the prism layer heights will be "floating" in order to produce a smooth volume transition to the tetrahedra (see Figure 271: All Prism Heights Floating (p. 296)). In order to smooth the volume transition, the last prism layer height must not exceed a certain height-to-base ratio. This value is 0.5 by default, but can be adjusted using the **Prism height limit factor**.

The **Prism height limit factor** combined with the shortest edge of the base triangle for each column are used to calculate the last prism height for each column. This is combined with the growth ratio, number of layers and the growth law to back calculate the initial height required for each column in order to produce a smooth growth ratio. If you would rather have a fixed initial height, you can redistribute the prism layers later.

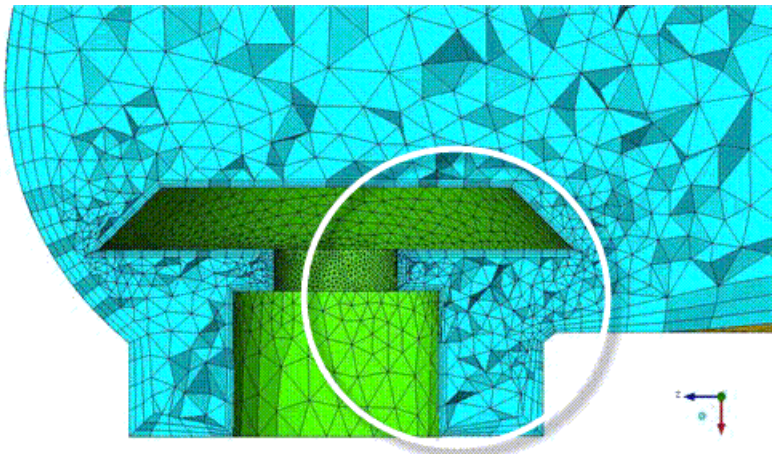
You can use a combination of floating prism layer height and prescribed initial/total height on selected entities/parts (see Figure 272: Combination of Prescribed and Floating Prism Heights (p. 297)). In this case, the initial height has been set on only the valve part. The prism heights on the valve part are now calculated using the specified initial height along with the growth law, ratio, and number of layers. All other prism heights remain floating.

Figure 271: All Prism Heights Floating



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 272: Combination of Prescribed and Floating Prism Heights



Interpolate Heights

if set to **On**, determines the prism layer heights of prism nodes without given heights by constructing new data points from prism nodes with fixed heights. If there are no fixed heights, the size of the triangle is used. By default, **Interpolate Heights** is off.

Compute params

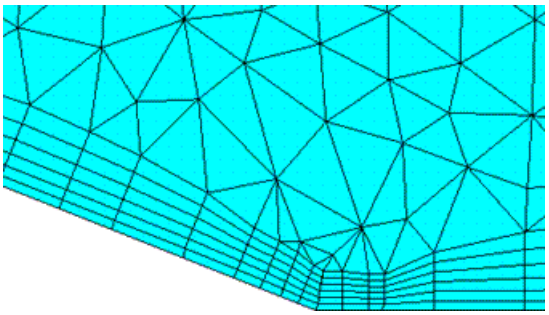
will compute the missing parameter based on the given parameters if one of the parameter values is blank.

Fix marching direction

if enabled, prisms are grown in a direction normal to the base triangles on the surface. However, the quality of mesh is still controlled by **Min prism quality**.

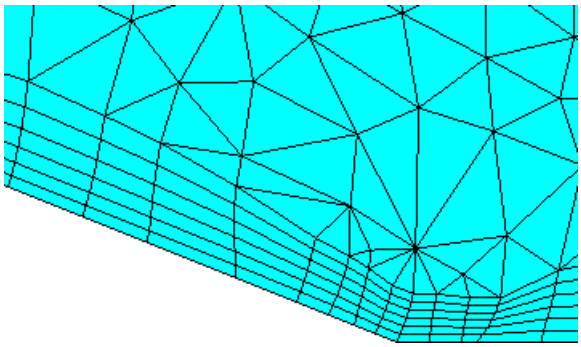
When **Fix marching direction** is enabled, directional smoothing applies only to the first layer. The initial direction is evaluated from the face normals attached to the vertex to be extruded. This is done by interpolation. For further layers, the direction is "fixed" and directional smoothing is disabled.

Figure 273: Use of the Fix Marching Direction Option



Fix Marching Direction Disabled

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

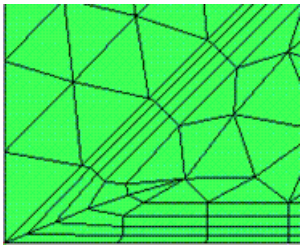
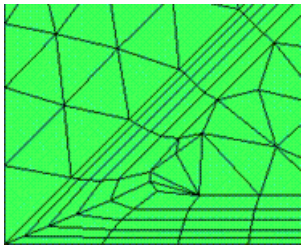


Fix Marching Direction Enabled

In the first example in Figure 273: Use of the Fix Marching Direction Option (p. 297), **Fix Marching Direction** is disabled and the prism direction is controlled by the directional smoothing steps and the **Ortho weight** value. In the second example, **Fix Marching Direction** is enabled. The prisms grow normal to the surface, except in cases of extremely poor quality first layer elements.

In the next set of examples in Figure 274: Fix Marching Direction Option (p. 298), prisms were grown largely orthogonal to the base triangles up to the 3rd layer, beyond which the **Min prism quality** caused a change in direction. This method may be more appropriate when growing prism in the VORFN region.

Figure 274: Fix Marching Direction Option

Fix Marching Direction Enabled	**Fix Marching Direction Disabled**
	

Note

If you do not use any directional smoothing, the prism marching direction is already fixed.

Min prism quality

locally prevents or adjusts prism growth in problem areas in order to maintain quality. Complex geometry combined with aggressive prism parameters may result in poor quality elements. If the quality does not meet the minimum value, the prism elements are re-smoothed directionally, or pyramids are used to replace some prism elements. Setting a value that is too high will result in local prism interruptions and more pyramids.

Table 8: Effect of Min Prism Quality and Initial Height (p. 298) demonstrates the effect of **Min prism quality** on prism mesh of two different initial heights.

Table 8: Effect of Min Prism Quality and Initial Height

Initial Height = 0.15, Growth Ratio = 1.2, Number of Layers = 5

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min prism quality value	Actual Prism Quality
0.0	0.003
0.01	0.065
0.075	0.1
0.1	0.13
Initial Height = 0.1, Growth Ratio = 1.2, Number of Layers = 5	
Min prism quality value	**Actual Prism Quality**
0.0	0.146
0.01	0.146
0.075	0.146
0.1	0.146

In contrast, for the mesh with an Initial Height of 0.1, the prism quality is already higher than the **Min prism quality** value.

Ortho weight

is the weighting factor for the directional smoothing steps. The range is from 0 (to improve triangle quality), to 1 (to improve prism orthogonality).

In the example in Figure 275: Ortho Weight = 0.1 (p. 299), the **Ortho weight** is set to 0.1. The smoother focuses on improving the aspect ration of the triangle caps, which also results in improved tetra quality.

Figure 275: Ortho Weight = 0.1

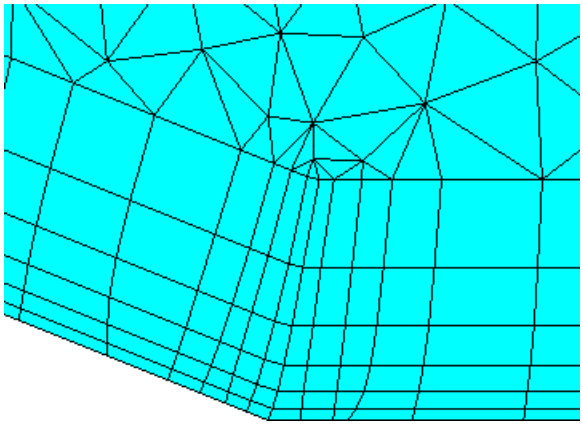


Figure 276: Ortho Weight = 0.5 (p. 300) illustrates the same mesh with **Ortho weight** = 0.5.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 276: Ortho Weight = 0.5

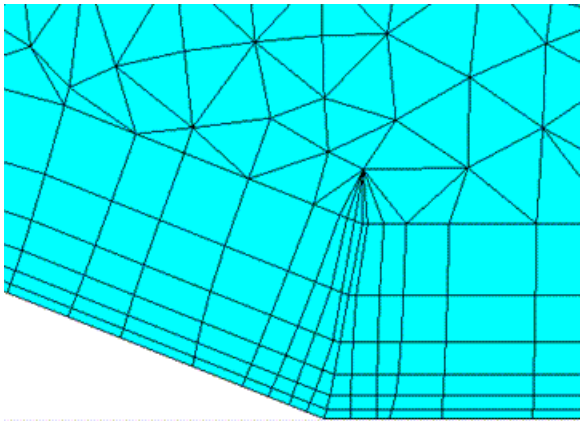
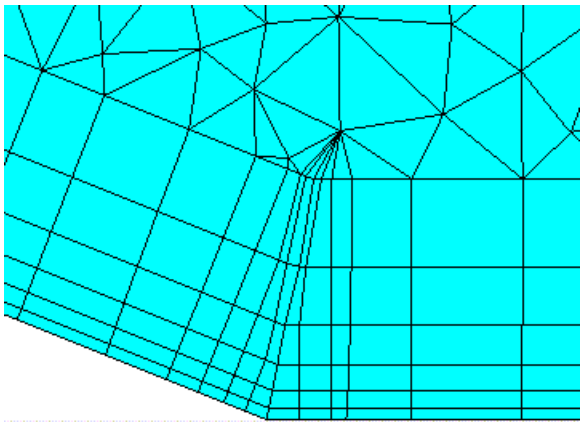


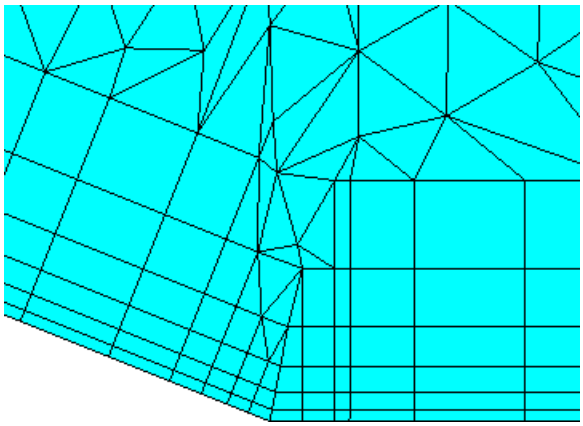
Figure 277: Ortho Weight = 0.9 (p. 300) illustrates the mesh with **Ortho weight** = 0.9. The smoother focuses on maintaining orthogonality with the wall, except when quality will be too poor.

Figure 277: Ortho Weight = 0.9



Ortho weight is applied during directional smoothing. If no directional smoothing is applied, the mesh is grown orthogonal to the wall in a fixed marching direction and poor elements are simply removed, as shown in Figure 278: No Directional Smoothing or Ortho Weight Applied (p. 300).

Figure 278: No Directional Smoothing or Ortho Weight Applied



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

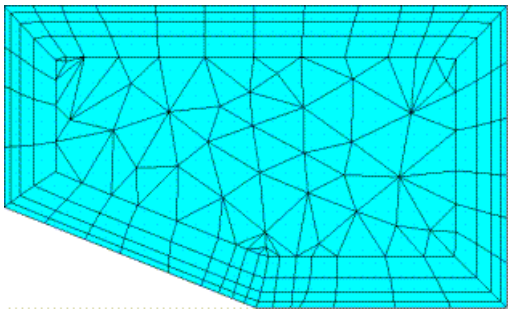
Fillet ratio

tries to create a fillet proportional to the total height of the prism. When prisms are grown in the corner zone of a tet mesh, it is possible to control the smoothness of prism layer along a fillet using this parameter. A value of 0 means no fillets.

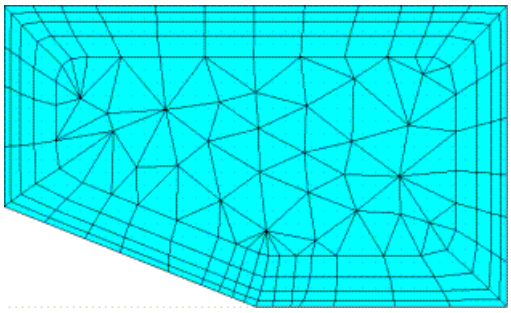
Note

For meshing corners with angles less than 60 degrees, there may not be space for a fillet.

Figure 279: Examples of Fillet Ratio

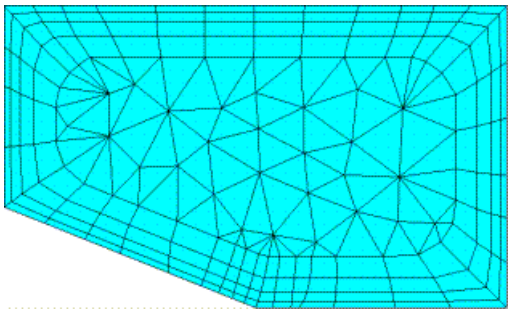


Fillet Ratio = 0.0



Fillet Ratio = 0.5

Here, the radius of the inner prism fillet is 0.5 times the height of the total prism thickness.



Fillet Ratio = 1.0

Max prism angle

controls prism layer growth around angles and when adhering to adjacent surfaces. This is the maximum internal angle between the base and the extruded direction. This parameter can range from 140 to 180 degrees. If extruding from one surface and not its neighbor, and the angle between the two surfaces is less than the specified value, the prisms will adhere to the adjacent wall.

For instance 135 is the theoretical minimum needed to go around a 90 degree corner (90+45 meeting up with the 90+45 coming from the other surface); to account for tolerances, the minimum setting is 140 degrees. A max prism angle of 180 allows the mesh to fold back on to its self, like at

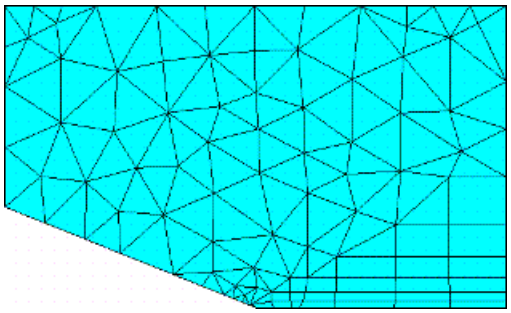
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the trailing edge of a wing or a cusp between pipes in a manifold (90+90 meeting up with the 90+90 from the other side).

Figure 280: Max Prism Angle – Example 1

Max Prism Angle = 140 degrees

Here, the angle between the planes is 158.2 (21.8) degrees. Since the **Max prism angle** is less than the angle between the walls, the prism layers are capped with pyramids.



Max Prism Angle = 180 degrees

Here, the **Max prism angle** exceeds the separation angle between the surfaces, so the prism remains attached to the adjacent surface.

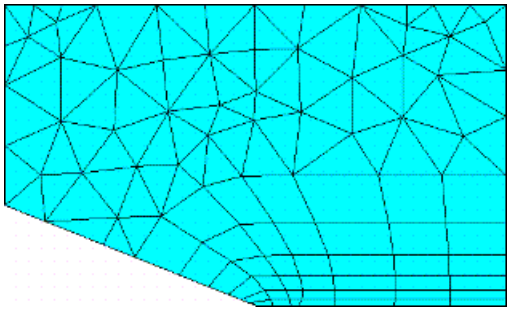
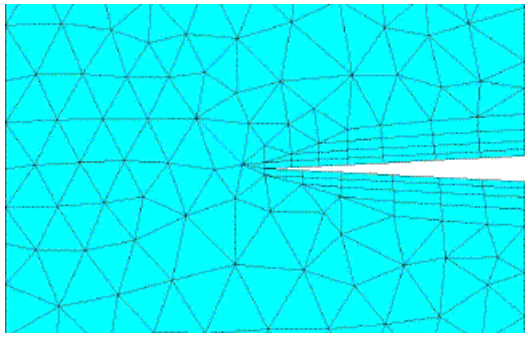


Figure 281: Max Prism Angle – Example 2

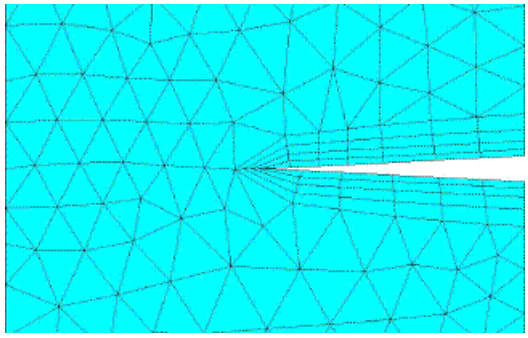
Max Prism Angle = 140 degrees

Here, the **Max prism angle** is set to 140, which is not enough to meet the prisms from the under side of the cusp.



Max Prism Angle = 180 degrees

Here, the **Max prism angle** is set to 180 and the **Min prism quality** is set to 0.0001 so that the prisms from the top surface can meet the prisms from the lower surface. The prisms where the two surfaces meet have large internal angles and poor quality.



Note

Pyramids are usually not favorable. However in “cusp” situations such that exist on the trailing edge of a wing or within a manifold where two pipes meet at a slight angle, it is better to have the prism layer terminate with pyramids than try to wrap

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

around the cusp. It is recommended to set **Max prism angle** to 160 degrees, particularly for applications with cusp geometry.

Max height over base

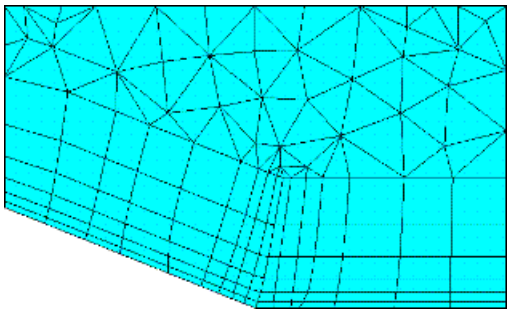
When the prism aspect ratio (ratio of height over base of the base triangle) reaches this value, the prism stops growing.

The examples in Figure 282: Max Height Over Base (p. 303) illustrate the use of this option for a 6 layer prism mesh.

Figure 282: Max Height Over Base

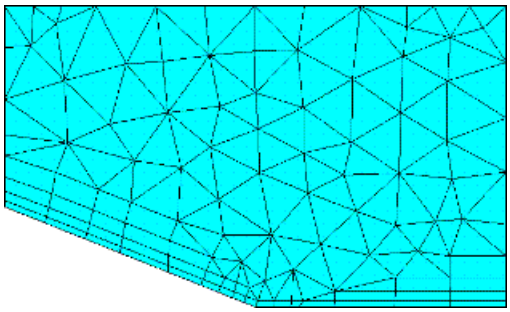
No Value Set for Max Height Over Base

In this case, the prisms with a smaller base have the same height as the prisms with wider bases. The resulting tall prisms have much greater volume than their adjacent tetra elements and are not considered ideal.



Max Height Over Base = 1.0

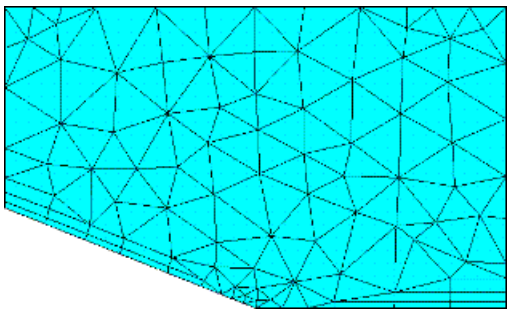
If the **Max height over base** is set to 1.0, then the prism height can not exceed the average base length. Prisms that would have exceeded this are not grown. Other prisms may not grow due to related quality issues.



Max Height Over Base = 0.5

In this case, even the initial height would exceed this ratio for the elements in the corner, so no prisms are grown there. Generally, the initial height at the wall would be less than half the smallest element size.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Prism height limit factor

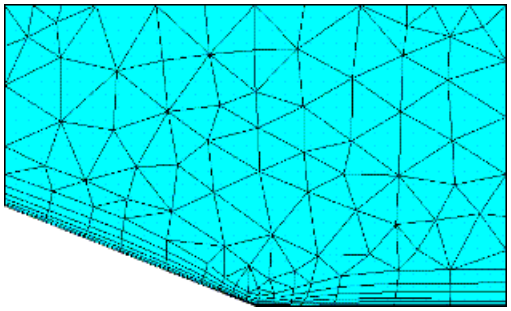
restricts the prism aspect ratio (height of each prism element over base size for each column) to the specified value by limiting the prism growth ratio, but maintains the number of layers.

If the initial height is set globally or on a part or entity, and the growing prism layers reach the specified aspect ratio, the growth ratio is reduced so that the height for additional layers will remain constant at that height limit factor (aspect ratio).

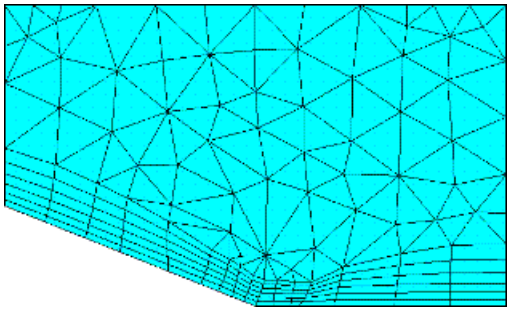
If the initial height is not set globally and not set on the part or entity, it will "float" (see the **Total height** section) to improve the transition to the tetrahedra. You can then use the prism height limit factor to control the transition by adjusting the maximum height-over-base ratio of the prisms. This will be used to calculate the initial heights on a column by column basis and so that a single growth ratio will be used and produce a smoother transition to the maximum prism height for each column. If the initial height is left to "float" and the prism height limit is not set, a value of 0.5 will be used to obtain smooth volume transition from the prisms to the attached tetra.

Figure 283: Prism Height Limit Factor

No Value Set for Prism Height Limit Factor

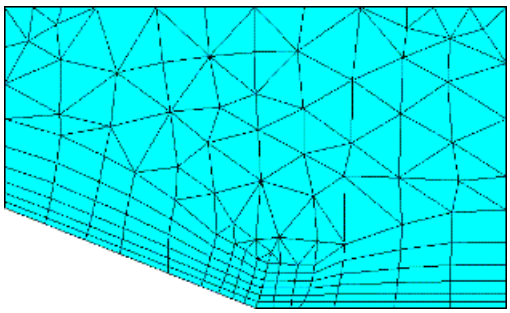


Prism Height Limit Factor = 0.5



Prism Height Limit Factor = 1.0

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ratio multiplier
only applies to the **Exponential Growth Law**. This value is multiplied to the height ratio for each successive layer. For example, if the **Height Ratio** is set to 1.2, and the **Ratio multiplier** set to 1.1, then the ratio between the first two layers will be 1.2, the ratio between the second and third layer will be (1.2 * 1.1), the ratio between the third and fourth layer will be (1.2 * 1.1 * 1.1), etc. The default value is 1. The **Ratio max** option under Advanced Prism Meshing sets the maximum ratio value.

Prism Element Part Controls

These settings are required when growing prism mesh without an existing tetra mesh. A part name is required to assign the prism elements correctly. This can be done by typing a new name or selecting from the list of parts.

You may assign a different volume part to the prism, or add a side or top part in order to better differentiate it from the tetra region. In this case, be sure to add the prisms to the main volume part before generating output to the solver so it will be seen as one volume.

You may want to grow prisms away from the flow domain (towards the outside region). In this case, the top triangles (Top Part), as well as side quads (Side Part) must also be assigned to named parts.

New volume part
allows you to specify a new part for prism elements, either from an existing surface or volume mesh.

Side part
allows you to specify a part for the quad faces on the side boundaries of the prism elements.

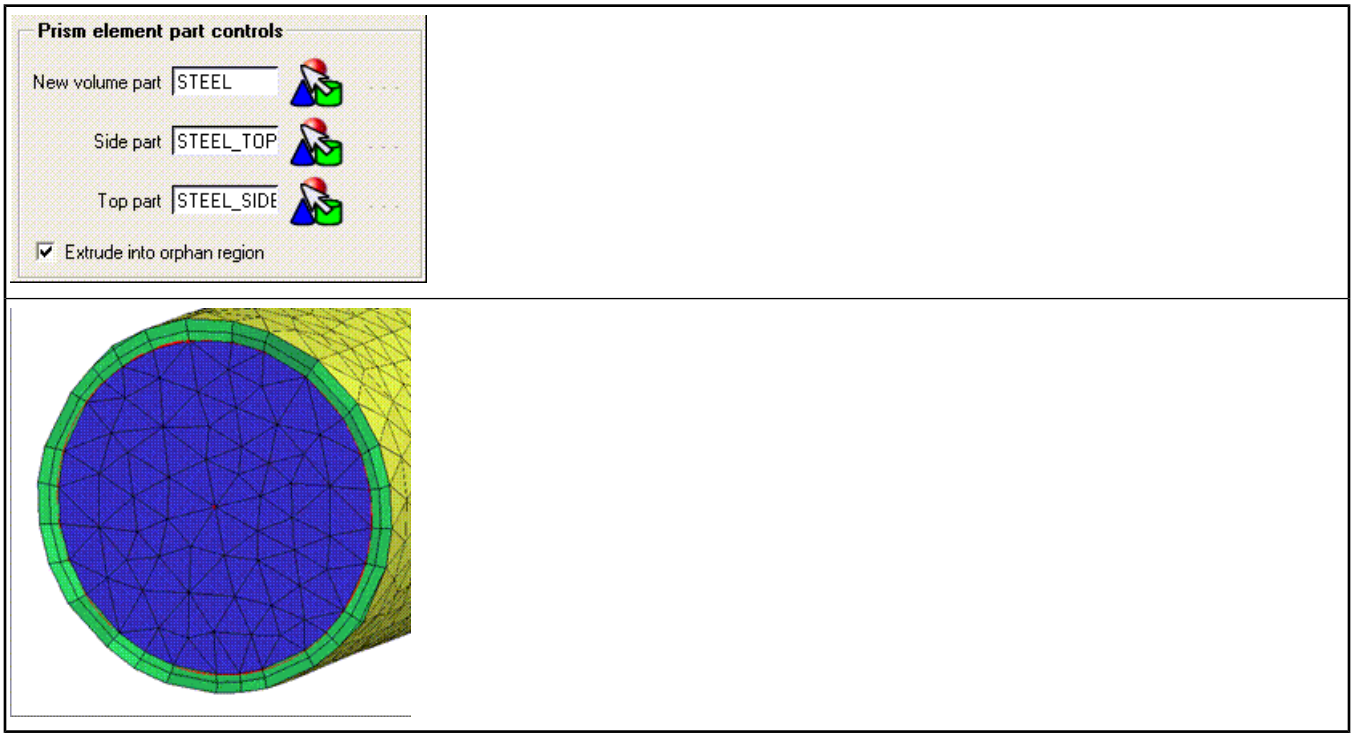
Top part
allows you to specify a part for the tri faces capping off the top of the last prism layer.

Extrude into orphan region
allows you to extrude prisms away from the existing volume, instead of into it.

In Figure 284: Prisms Extruded into the Orphan Region (p. 306), prism mesh was extruded from the pipe mesh region. The part STEEL_SIDE represents the metal's thickness.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 284: Prisms Extruded into the Orphan Region



Smoothing Options

Number of surface smoothing steps

is the number of iterations that the surface mesh is smoothed before prism layer generation. The quality of the final prism mesh mostly depends on triangle quality. It is recommended to start with quality above 0.3.

Note

Set smoothing steps to 0 while extruding only one layer. Otherwise the default value is usually adequate.

Triangle quality type

specifies various quality criteria for improving the triangular surface mesh by smoothing. This operation is performed at the beginning as well as before extruding a new layer.

When **Laplace** is selected, the mesh is improved using the Laplace elliptic solver. Separate weighting is given for orthogonality and triangle quality.

Note

Laplace smoothing is typically best for final prism quality.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of volume smoothing steps

is the specified number of smoothing steps on the existing Tetra mesh before prism layer generation. A smooth Tetra mesh is essential for a high quality prism mesh. Smooth the Tetra mesh and perform all diagnostics before running this smoother, in which case you may reduce the number of iterations.

Note

The smoother is much more effective before prism mesh is grown.

Note

If extruding only one layer, set smoothing steps to 0. Otherwise the default value is usually adequate.

Max directional smoothing steps

is the number of smoothing steps for the face normal vectors before the next prism layer is created. The default value is appropriate for most problems.

First layer smoothing steps

smooths the first layer with the given number of steps.

Advanced Prism Meshing Parameters

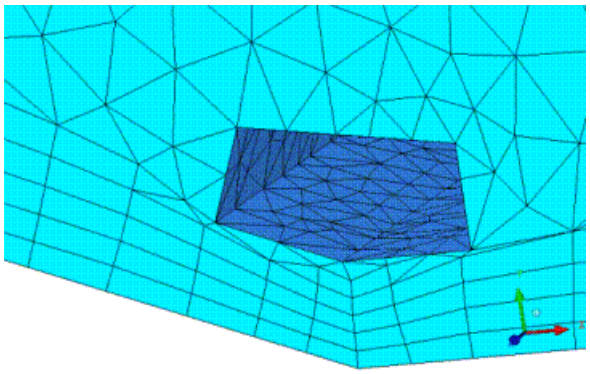
Auto Reduction

In some cases, the prism mesher is unable to create the specified number and size of layers due to geometry constraints. If this option is enabled, the prism mesher will automatically reduce the layer size to meet the required number of layers without creating pyramids or other problems in the mesh.

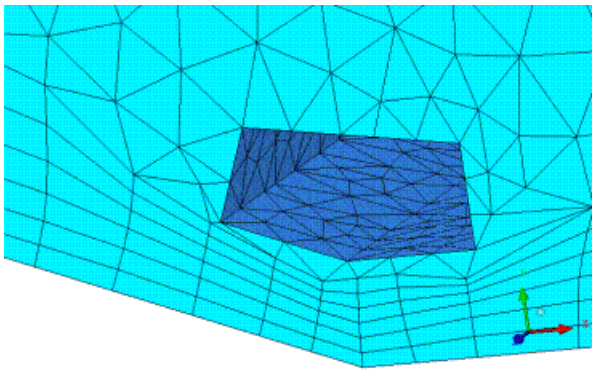
This is useful for solvers that require complete prism layers with no pyramids. In Figure 285: Example of Auto Reduction (p. 307), the prism parameters were set for 6 layers. Without **Auto Reduction**, there will be stair stepping down to 4 layers in the gap. With **Auto Reduction** enabled, the mesher created 6 layers as specified by automatically reducing the layer size to fit and redistributing the prism layer as appropriate to improve quality.

Figure 285: Example of Auto Reduction

Auto Reduction disabled



Auto Reduction enabled



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blayer 2d

This option is used to create a boundary layer in a 2D mesh using prism meshing. This surface mesh can be used with a 2D solver or swept to create the volume mesh.

Note

Using Prism BLayer 2D is more advanced than applying an Offset with the Patch Independent meshers. Offset is controlled by setting the Height, Ratio and Number of layers parameters on the curves. 2D prism controls are the same as regular prism controls, except you must select the surface part as well as the curve part. 2D Prism can also be applied on an existing shell mesh in the same way that 3D prism is applied to an existing tetra mesh.

Note

Since 2D prisms are really just quads, the split prism and redistribute prism commands do not work.

2D prism works with quad or tri mesh and can be controlled with the 3D prism controls to adjust number of layers, initial height, orthogonality, etc.

Figure 286: BLayer 2D Applied to a 2D Surface with Quad Mesh

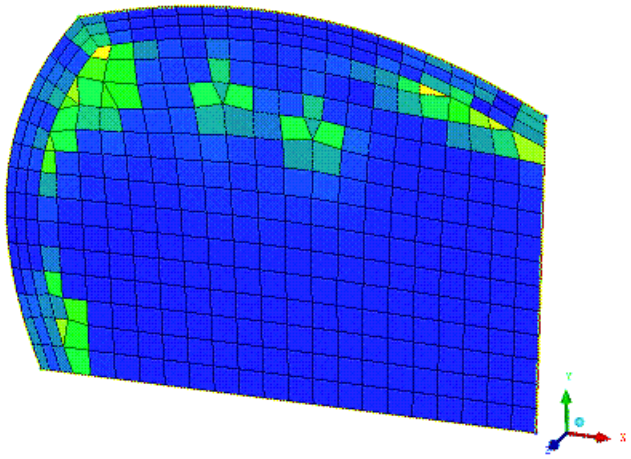


Figure 286: BLayer 2D Applied to a 2D Surface with Quad Mesh (p. 308) shows a 2D surface with quad mesh, with prism applied to the top and left curves.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 287: BLayer 2D Applied to a 2D Surface with Tri Mesh

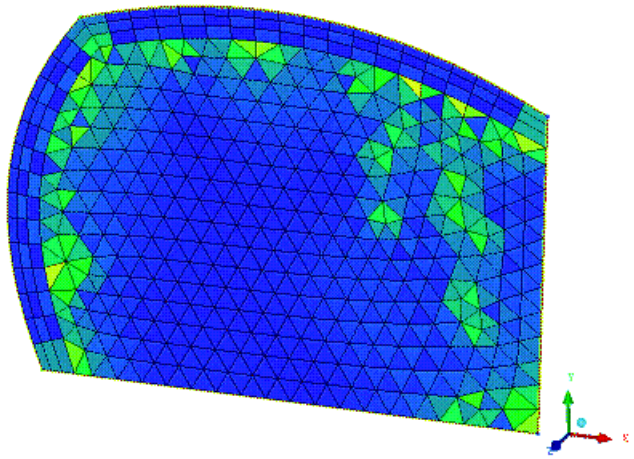


Figure 287: BLayer 2D Applied to a 2D Surface with Tri Mesh (p. 309) shows a 2D surface with tri mesh, with prism applied to the top and left curves. Note that the initial height is "floating" in this case.

Figure 288: BLayer 2D and Additional Prism Parameters Applied to a 2D Surface with Tri Mesh

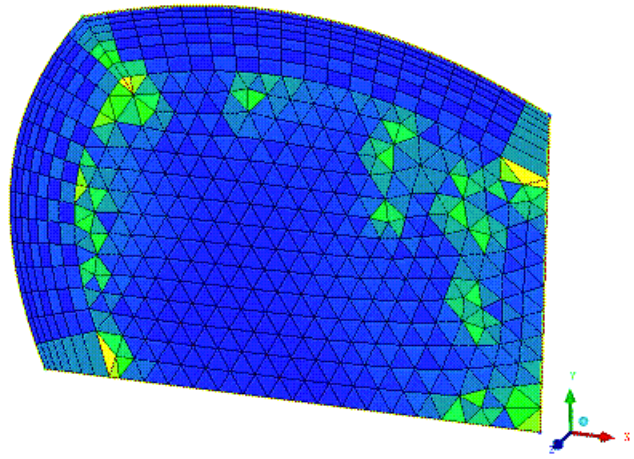


Figure 288: BLayer 2D and Additional Prism Parameters Applied to a 2D Surface with Tri Mesh (p. 309) shows the same 2D surface with tri mesh, with prism initial height set to 1, ortho weight set to 0.5, and number of layers set to 7.

Delete Base Triangles

If enabled, the triangles at the interface with the existing tetra elements will be deleted when growing into ORFN regions.

Delete Standalone

If enabled, triangles that are not attached to anything will be deleted.

Do Checks

If enabled, additional checks are done on the mesh. These include checking interfaces between materials and checking for pockets within the mesh. Results are reported in the message window and the prism log file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Do Not Allow Sticking

If enabled, "pointy" tetrahedra are not allowed to be stuck between layers.

Intermediate Write

If enabled, the prism file will be written out after each layer is created. This is a fail safe measure. If the Prism mesher is not able to complete all the layers or is interrupted for some reason, you can load the saved mesh file and continue. The mesh file will be located in your working directory and named "prism.uns".

Refine Prism Boundary

If enabled, this allows the Prism mesher to refine the boundary between the prisms and tetras, if this will improve the mesh quality.

Stair Step

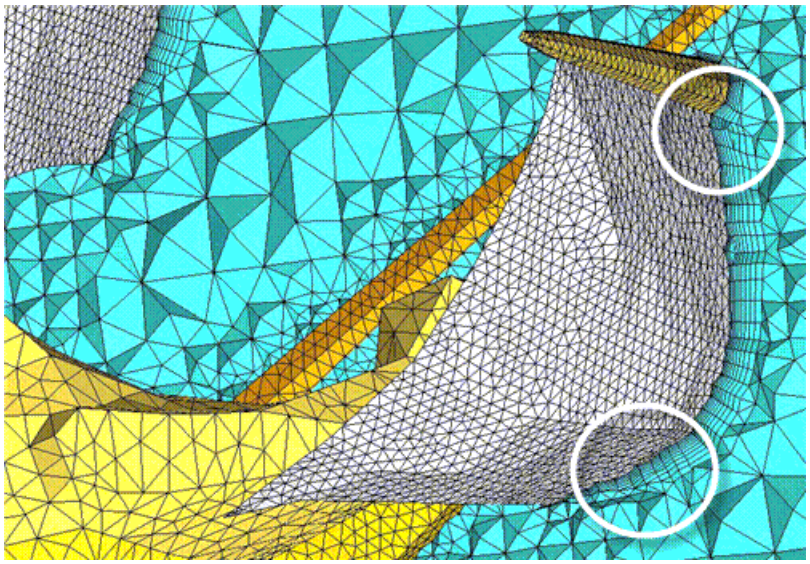
This option affects the behavior of the Stop Columns option, and is only effective if **Stop Columns** is enabled.

If **Stair Step** is enabled, the difference in adjacent prism columns is limited to 1 layer. When disabled, adjacent columns can have different numbers of layers that differ by more than 1.

Figure 289: Using the Stair Step Option (p. 310) shows the prism layers grown on a ship sail. Note that when the **Stair Step** option is disabled, the prism layers are fully extruded on the sail.

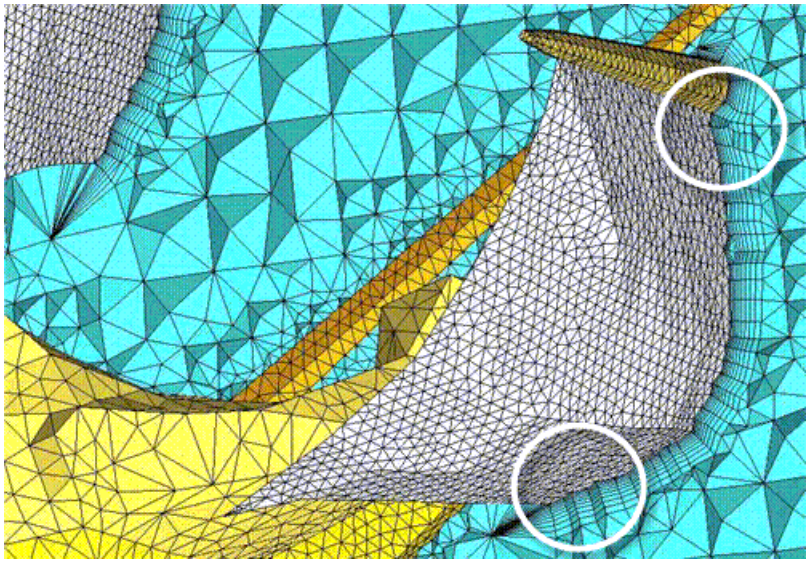
Figure 289: Using the Stair Step Option

Stair Step option enabled



Stair Step option disabled

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

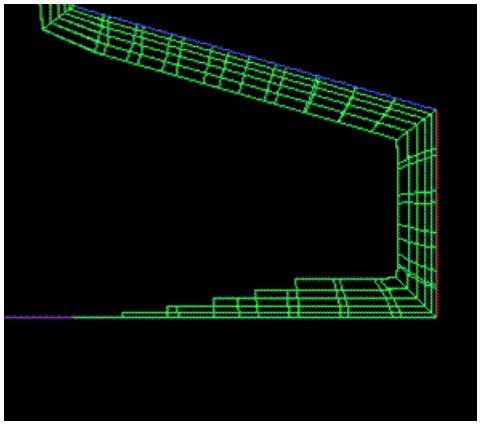


Stop Columns

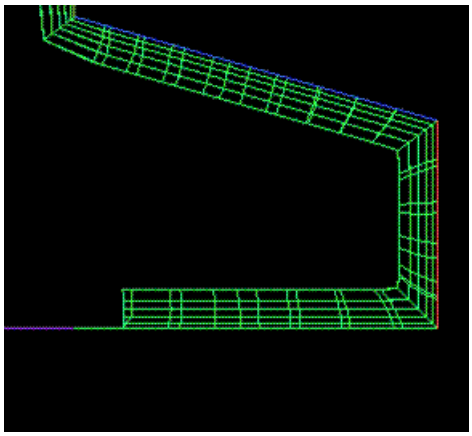
If enabled, a column of prisms is stopped if the marching direction cannot be determined or a collision is detected. If disabled, and a collision is detected at any point, the entire next layer is stopped unless **Auto Reduction** is also enabled. In many cases, disabling this option may cause the Prism mesher to terminate prematurely.

Figure 290: Example of Stop Columns and Stair Step Options

Stair Step option enabled:



Stair Step option disabled:



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Existing Quad Layers
This option deals with cases where a prism layer meets a wall. If enabled, the software will attempt to align the boundary of the prism layers to the existing quad surface mesh. If the existing mesh has a different structure or the prism mesh has trouble mapping to the structure, then the prism faces will be used for the surface mesh. If disabled, this option will adjust the existing mesh to follow the faces of the uncovered prisms.

Use Prism V10
uses the ANSYS ICEM CFD Version 10 prism executable (prism_v10.exe) instead of the default executable.

Max Jump Factor
The maximum allowable jump in height between neighboring vertices (default is 5.0). If adjacent prism columns exceed this ratio, the Prism mesher will terminate prematurely and an error message will note that Max Jump Factor was exceeded.

Max Size Ratio
The maximum allowable size ratio when growing larger prisms into smaller tetra elements (default is 0.0).

Min Smoothing Steps
The minimum number of marching direction smoothing steps (default is 6). This does not override **Max directional smoothing steps**, but acts as a lower limit.

Min Smoothing Val
The marching directions will be smoothed until the quality is above this value (default is 0.1) or the **Max directional smoothing steps** is reached. If **Min Smoothing Val** is met, but the **Max directional smoothing steps** has not been yet reached, smoothing will continue until the number of **Min Smoothing Steps** have been completed.

Ratio Max
The maximum height ratio between prisms from one layer to the next.

Tetra Smooth Limit
The tetra elements will be smoothed until they are above this value (default is 0.3).

Read a Prism Parameters File
allows you to select an existing prism parameters file. This will be used to set all the prism parameters, including Advanced Prism Meshing Parameters. These prism parameters files can be shared between projects for consistency and easy setup.

Periodicity Set Up

The **Periodicity Set Up** option allows you to define periodicity.

Define Periodicity
ensures that the nodes line up from one periodic face to another.

Note

Behind the interface, periodicity is always defined in the global coordinate system. However, it can be setup with an active local coordinate system (LCS). When you click

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply, the axis vector will be normalized and the LCS parameters will be converted to global coordinates.

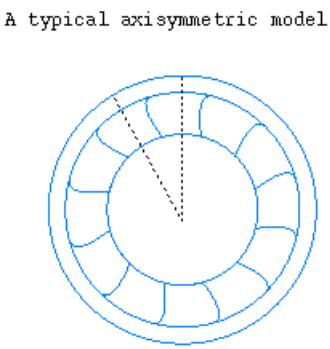
Note

Periodicity can be set for both **Tetra** and **Hexa** meshes.

Rotational periodic

This option ensures that the nodes will line up along an axis symmetric model, and also force the nodes to be rotationally periodic with one another. An example of rotational periodicity is that a node and its periodic counterpart on an opposing periodic face share the same R and Z coordinates of a cylindrical coordinate system. **ANSYS ICEM CFD** has the ability to mesh all the way to the axis of rotation.

Figure 291: Rotational Periodic Geometry



There are three methods to defining the Rotational axis.

The **User defined** methods require the following parameters:

- Base

  The base point of rotation (x y z).

- Axis

  A vector defining the direction of the axis of rotation (i j k).

- Angle

  The angle that makes up the computational domain within the axis symmetric model. An angle of 360 degrees would mesh the entire rotational domain.

  OR

  Sectors

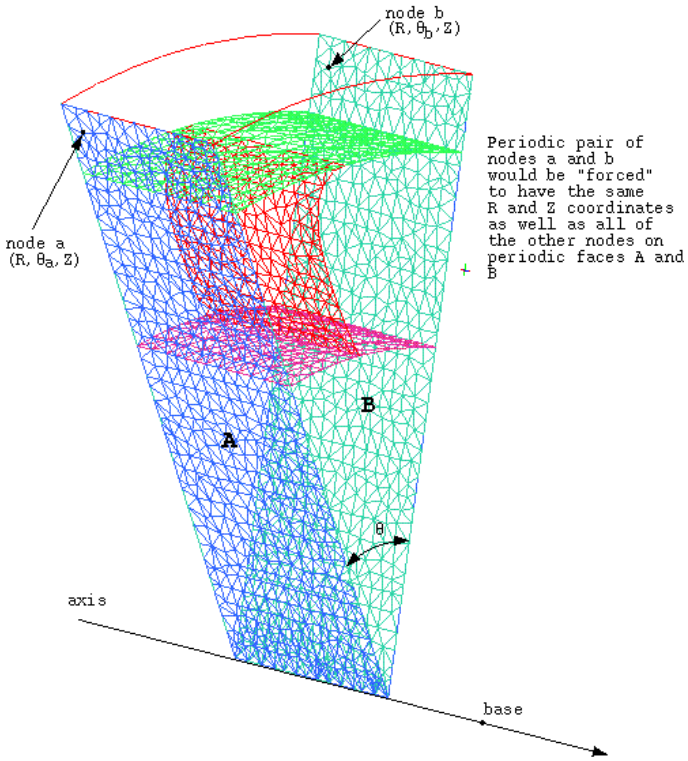
  The number of sectors in 360 degrees that make up the entire rotational computational domain.

The **Vector** method requires 2 points to define the vector for the axis of rotation, and the angle to define the rotational domain.

An example of a rotational periodic mesh is shown in Figure 292: Rotational Periodic Mesh (p. 314).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

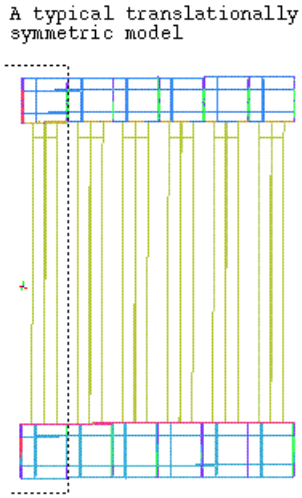
Figure 292: Rotational Periodic Mesh



Translational Periodicity

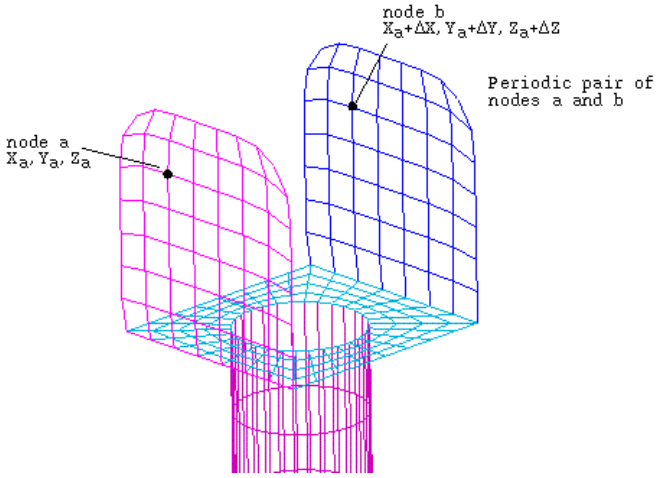
This option ensures that the nodes line up along a translational symmetric model, as illustrated in Figure 293: Translational Periodic Geometry (p. 314) and Figure 294: Translational Periodic Mesh (p. 315). Enter the **Offset** vector value that defines one periodic face from the other (Dx Dy Dz).

Figure 293: Translational Periodic Geometry



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 294: Translational Periodic Mesh



Part Mesh Setup

The **Part Mesh Setup** option opens a dialog where you can specify the mesh parameters for different parts, as shown in Figure 295: Part Mesh Setup Window (p. 315). Values defined here at the Part level will override Global settings. If a value of 0 is entered for a parameter, the global parameter value will be used.

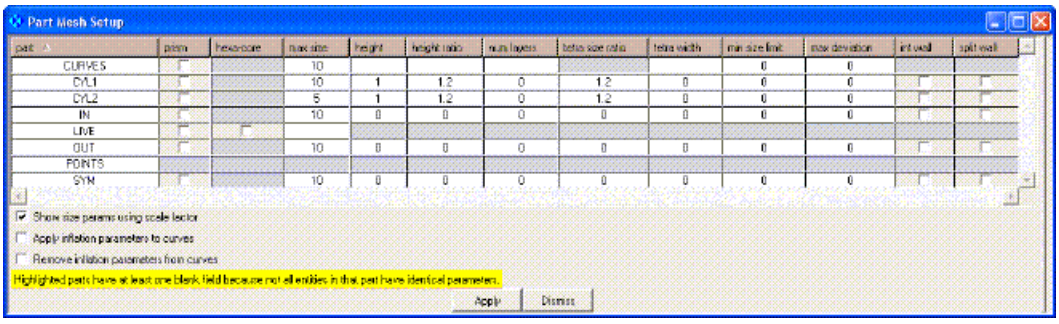
Note

Entity level controls will override Part level parameter settings.

Note

Clicking the header cell for a parameter (e.g.,**max size**) will open up a window, which allows you to specify the same parameter value for all available parts.

Figure 295: Part Mesh Setup Window



Prism

Select the parts that prism layers will be applied to. For each part, the Height, Height Ratio and the Num Layers can be specified. Volume, Surface and/or Curve parts can be selected. When using the Prism mesher in ANSYS ICEM CFD, the default prism parameters are set under Mesh > Global Mesh Setup > Global Prism Settings. However, parameters that are entered here at the Part level will override the Global settings. In addition, the entity settings of Height, Height Ratio, and Number of

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layers will override the Part level settings. Entering zero for any of the settings will default back to the Global Settings.

If there are multiple volume materials, and none have the **prism** option enabled, then prism mesh will grow from the selected surface parts into the adjacent volumes. If only certain volume parts have **prism** enabled, then prism mesh will be grown only into those volume parts. After you enter the parameters, click **Apply**, and then **Dismiss** to close the window.

For 3D, the prisms are grown (inflated) from the shell (tri or quad) elements of each part. This can be done with or without a volume mesh, but having a tetra volume mesh during inflation helps with collision avoidance and ensures that you will have a volume mesh after prism generation is complete. For 2D, the prisms are grown (inflated) from the curve parts into the selected surface parts (must be selected). The 2D prism only works if Advanced Prism Meshing Parameters > Blayer 2D is enabled.

Note

At least one surface entity must be selected or the prism mesher will not run. Different prism heights can be specified on adjacent parts, though a transition region with unspecified height is required in between these parts. If a surface part separates two or more volume parts, select the volume parts on the side of the surface you want to grow the prisms. If you select both sides, prisms will grow in both directions from the surface part. If you want different prism properties on either side of a surface grown into 2 volumes, do one at a time (run prism iteratively). If adjacent tri element parts have heights that differ by more than a factor of 2, the prism mesher may fail (this limit is controlled in the **Advanced Prism Meshing Parameters**). Not setting a height for Prism, (here or in the **Global Prism Parameters** DEZ) will allow the height to float. You can also allow the height to float globally and set specific initial heights per part or on an entity by entity basis.

Hexa-Core

Enable this option to use Hexa-Core meshing for the volume part, and to set the desired parameters. The global parameters for Hexa-Core meshing can be set under Mesh > Global Mesh Setup > Volume Meshing Parameters > Cartesian Mesh Type > Hexa-Core Mesh Method. The default Max Size is based on a weighted average of the perimeter surface elements.

If **all** or **none** are selected, Hexa-Core will fill all volumes (similar to Prism).

To enable Hexa-Core meshing for all available parts, click the hexa-core header and all available parts will be selected.

Max Size

specifies the maximum element size. The actual maximum element size will be this value multiplied by the **Global Element Scale Factor**.

Note

Local maximum element sizes defined on bodies/materials can be used to limit the maximum size for the Hexa-Core method. For other methods, such as BFCart, Octree Tetra or Delaunay Tetra, you will need to set the local maximum size using a density region (refer to Create Mesh Density (p. 327) for details). This is because the material is applied (via flood-fill from the material point) after subdivision is complete.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Height

specifies the height of the first layer of elements normal to the surface or curve. For volume meshing, this parameter affects the Hexa and Prism initial mesh layer height. For Patch Dependent Surface meshing, when applied to a curve, this value can affect the initial height of the layer of quads along that curve. For example, this could be used to specify the initial height of a quad ring around a bolt hole.

Height Ratio

is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the next layer.

The default growth rate for the transition to the surface Max Size is 1.5. This growth rate can be adjusted by setting the surface Height ratio to between 1.0 and 3. Sizes below 1.0 are inverted (i.e., 0.667 = 1.5). Sizes above 3 are ignored and the default is used.

When applied to curves, the Height Ratio can have several effects on Patch Dependent meshing. When used with an initial Height and Number of Layers, it determines the growth rate of one layer of quads over the previous layer. When used with the **Adapt Mesh Interior** setting, it affects how quickly the mesh transitions from the curve sizes to the surface sizes.

Num Layers

is the number of layers to be grown from the surface or curve.

Tetra Size Ratio

controls the growth (edge length) of tetra mesh as it moves away from the surface.

For Octree Tetra mesh, size transitions are 2 to 1 locally, but you can still affect the transition rate over a larger number of transitions. For example, if the surface size is 2 and the volume size is 64, you may see something size transitions like the following:

Delaunay	2	3	4.5	6.75	10.13	15.19	22.78	34.17	51.26	76.89
Octree	2	2	4	4	8	8	16	32	32	64

If the ratio was 1.5, the big picture would show both the Octree and Delaunay sizes growing at a rate of 1.5, but the Octree method has to fit it to powers of 2 times the smallest size in the model. Also, you can imagine that changing this to 1.4 or 1.6 might not make that much difference to the Octree mesher, at least not for the first few layers, but would directly change the Delaunay progression.

Tetra Width

creates the specified number of tetra layers with element size as specified by the **Max Size**.

Min size limit

Mesh elements will be limited from being subdivided smaller than this value. For Tetra meshing, this value overrides the value calculated by the **Curvature/Proximity Based Refinement** option. The actual minimum size will be the value multiplied by the **Global Element Scale Factor**.

This parameter works only with the **Curvature/Proximity Based Refinement** option. You can override the global setting on particular entities or parts if this is set smaller than the global setting. Min size limit functions to limit the amount of **Curvature/Proximity Based Refinement**, however, if the sizing function is satisfied without refining down to this minimum size, it will stop on its own and may never refine down to this Min size limit value.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Deviation
is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Int Wall
if enabled, the part will be meshed as an internal wall. This applies to Octree Tetra Meshing only. This is necessary if you wish to mesh the surface of an internal wall within a volume.

Split Wall
if enabled, the part will be meshed as a split wall with overlapping pairs of elements and nodes, so that both sides of the wall are effectively treated as having surface elements. This applies to Octree Tetra Meshing only.

Show size params using scale factor
adjusts the reference element display to show the actual **Max Size** on each entity after multiplying the set maximum size by the **Global Element Scale Factor**. To display this size for each entity, use the display options in the Display Tree, such as the **Tetra Sizes** or **Hexa Sizes** options under **Surfaces**.

Apply inflation parameters to curves
allows you to apply the inflation parameters (Height, Height ratio, and Num. layers) to curves. By default, these parameters are applied only to surfaces because these options can produce a different result with the Patch Dependent Surface mesher when applied to curves. For instance, if you intend for these parameters to affect only Prism generation (volume mesh), they should not be applied to the curves if you also intend to use the Patch dependent surface mesher and do not want the surface mesh to be affected.

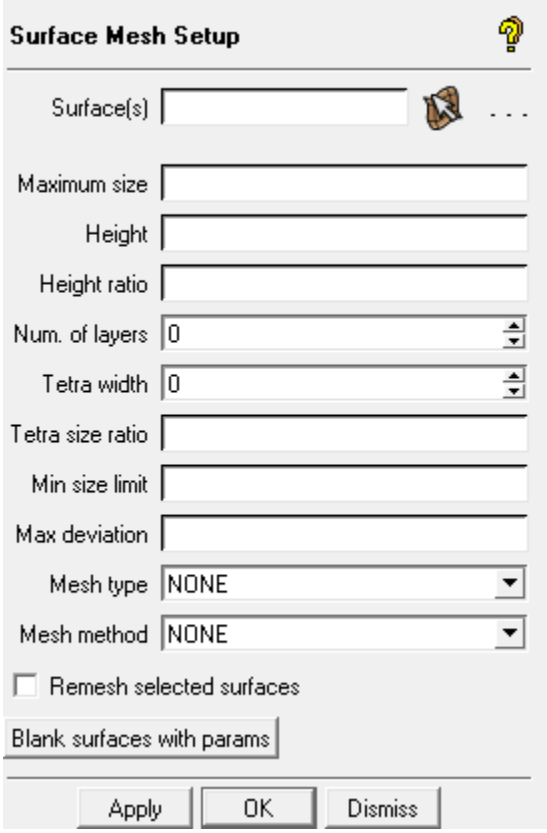
Remove inflation parameters from curves
allows you to remove any parameters related to inflation layers from curves.

Surface Mesh Setup

The **Surface Mesh Setup** option allows you to set mesh parameters for selected surfaces. For models with blocking, this applies only to 3D blocking.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 296: Surface Mesh Setup



Surface(s)
specifies the surfaces for which meshing parameters are to be defined.

Maximum size
specifies the maximum element size. The actual size is this value multiplied by the **Global Element Scale Factor**. You may end up with small elements on the selected entities if you are using **Curvature/Proximity Based Refinement** or **Maximum Deviation**.

Height
is the height of the first element, normal to the entity. This value applies to Hexa mesh only, and can be any positive real number.

Height ratio
is the expansion ratio from the first layer of elements on the surface. This ratio will be multiplied by the element height of the previous layer to define the next layer. This value is used for Prism and Hexa mesh, and can be any positive real number.

The default growth rate for the transition to the surface Max Size is 1.5. This growth rate can be adjusted by setting the surface Height ratio to between 1.0 and 3. Sizes below 1.0 are inverted (i.e., 0.667 = 1.5). Sizes above 3 are ignored and the default is used.

Number of layers
is the number of layers to be grown from the surface or curve.

Tetra width
creates the specified number of tetra layers with element size as specified by the **Max size**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tetra size ratio
controls the growth of tetra as it move away from the surface. It is used for Tetra mesh.

Min size limit
Mesh elements will be limited from being subdivided smaller than this value. This option is the same as **Curvature/Proximity Based Refinement**, but is applied locally to the selected part or entities. This is a factor that is multiplied by the **Global Element Scale Factor**.

This parameter works only with the **Curvature/Proximity Based Refinement** option. You can override the global setting on particular entities or parts if this is set smaller than the global setting. Min size limit functions to limit the amount of **Curvature/Proximity Based Refinement**, however, if the sizing function is satisfied without refining down to this minimum size, it will stop on its own and may never refine down to this Min size limit value.

Maximum deviation
is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Note

This only applies to the interior of surface mesh, not to the boundary. Node bunching settings for the boundary curves will still be respected.

Mesh type
specifies the mesh type for the selected surface. If a mesh type is selected for a specific surface, then this will override the global mesh settings. See Global Mesh Setup > Shell Meshing Parameters for a description of mesh types.

Mesh method
specifies the mesh method for the selected surface. If a mesh method is selected for a specific surface, then this will override the global mesh settings. See Global Mesh Setup > Shell Meshing Parameters for a description of Mesh methods.

Remesh selected surfaces
allows you to remesh selected surfaces after changing surface mesh parameters. The new surface mesh will automatically be generated.

Blank surfaces with params
when toggled, the surfaces with parameters applied to them will be make invisible or visible.

Curve Mesh Setup

The **Curve Mesh Setup** option allows you to set mesh parameters for curves. This feature is useful for both surface and tetra meshing in controlling mesh size or distribution on surface boundaries. There are several parameters, but in most cases setting the element size is sufficient for meshing.

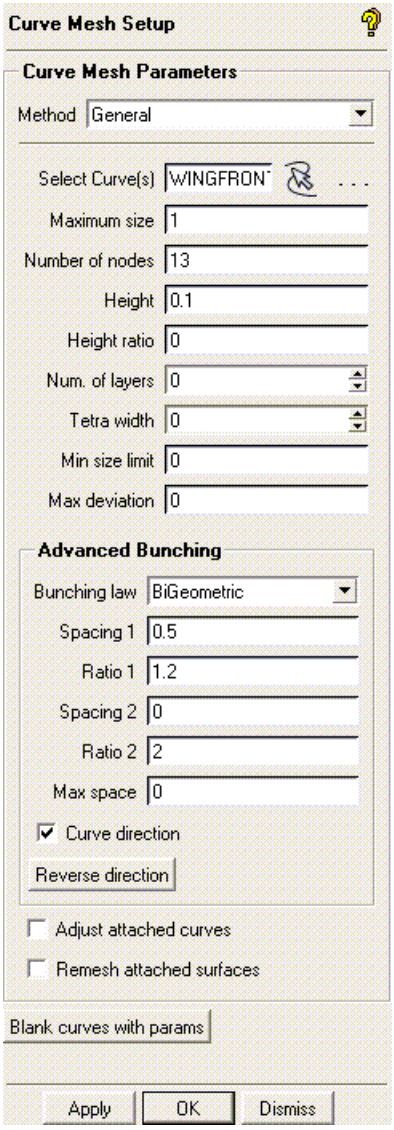
There are the following methods for setting the Curve Mesh Size:

- Figure 297: Curve Mesh Setup – General (p. 321): This method involves entering the parameter values in the provided fields.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Figure 298: Curve Mesh Setup – Dynamic (p. 325): This method allows you to select a curve and adjust the values using the mouse buttons.
- Figure 299: Curve Mesh Setup – Copy Parameters (p. 326): This method allows you to copy curve mesh parameters to the selected curve(s).

Figure 297: Curve Mesh Setup – General



General

allows you to enter the following parameter values in the respective fields:

Select Curve(s)

specifies the curves for which mesh parameters are to be defined.

Maximum size

is the maximum element size for the selected curves. The actual element size will be this value multiplied by the **Global Element Scale Factor**. The element size can be displayed by right clicking on **Curves** in the Model tree and selecting **Curve Tetra Sizes** or **Curve Hexa Sizes**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of nodes
is the number of elements for the selected curves. This is an alternative option to setting the element size. The number of elements entered should be more than 2. The node spacing can be displayed by right clicking on **Curves** in the Model tree and selecting **Curve Node Spacing**.

Height
is the height of the first elements, normal to the curve. This value applies to Hexa mesh and Patch Dependent Surface mesh, and can be any positive real number.

Height Ratio
is an expansion ratio from the first layer of elements on the curve. This ratio will be multiplied by the element height of the previous layer to define the next layer. You can enter any positive real number. This parameter applies to Hexa, Prism mesh, and Patch Dependent Surface mesh.

Number of layers
is the number of layers to be grown from the surface or curve.

Tetra width
creates the specified number of tetra layers with element size as specified by the **Max size**.

Min size limit
Mesh elements will be limited from being subdivided smaller than this value. For Tetra meshing, this value overrides the value calculated by the **Curvature/Proximity Based Refinement** option. The actual minimum size will be the value multiplied by the **Global Element Scale Factor**.

This parameter works only with the **Curvature/Proximity Based Refinement** option. You can override the global setting on particular entities or parts if this is set smaller than the global setting. Min size limit functions to limit the amount of **Curvature/Proximity Based Refinement**, however, if the sizing function is satisfied without refining down to this minimum size, it will stop on its own and may never refine down to this Min size limit value.

Maximum deviation
is a method of subdivision based on the proximity of the centroid of a tri or quad surface element to the actual geometry. If the distance is greater than this value, the element will automatically split and the new nodes will be projected onto the geometry. The actual distance is the value multiplied by the **Global Element Scale Factor**.

Advanced Bunching
contains options providing more control over the mesh parameters. Refer to Blocking > Pre-Mesh Params > Edge Params > Bunching Laws for more detailed explanation.

Bunching law

BiGeometric
This is the default bunching law. The two initial heights and ratios define parabolas in a co-ordinate system where the number of node points is the X-axis and the cumulative distance along the edge is the Y-axis. The parabolas are truncated where their tangent lines are identical; the spacing is linear between these points. If there are not enough nodal points to form this linear segment, a hyperbolic law is used and the ratios are ignored.

Biexponential
The spacing of the node intervals is calculated according to the law specified for Exponential 1 and 2. However, Spacing 1, Ratio 1, Spacing 2 , and Ratio 2 are used to define the distribution. The Spacing 1 and Ratio 1 parameters define the distribution from the beginning of the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

edge to the midpoint of the edge, and Spacing 2 and Ratio 2 define the distribution from the terminating end of the edge to the midpoint of the edge.

Curvature
The spacing of the node intervals is calculated according to the curvature of the function defining the distribution.

Exponential 1
The spacing of the i'th interval is defined as a function of the Spacing 1 and Ratio parameters.

Exponential 2
The same as Exponential 1, except that the Spacing 2 and Ratio 2 parameters are used and the distribution starting point is the terminating end of the edge.

FullCosinus
The spacing of node intervals is calculated using the cosine function. The ends of the edge have the same constraint values for spacing and ratio.

Geometric 1
Spacing 1 is used to set the first distance from the starting end of the edge, with the remaining nodes are spaced with a constant growth ratio. Only Spacing 1 is specified.

Geometric 2
The same as Geometric 1, except that Spacing 2 is used to define the distribution starting from the terminating end of the edge.

HalfCosinus1
The spacing pattern follows a half cycle of a Cosine function. The parameters for spacing and ratio differ on either end.

HalfCosinus2
The spacing pattern follows a half cycle of a Cosine function. The parameters for spacing and ratio are similar to HalfCosinus 1, but distribution starts from the opposite end.

Hyperbolic
The spacing at each end are used to define a hyperbolic distribution of the nodes along the edge. You can set Spacing 1 and Spacing 2, and the growth ratios are determined internally.

Linear
The spacing of the node intervals is calculated using a linear function.

Poisson
The spacing of the node intervals is calculated according to a Poisson distribution. Requested values of Spacing 1 and Spacing 2 are used. Requested values of Ratio 1 and Ratio 2 are not used directly, but are used to determine if the requested spacing is appropriate. If not, the spacing will be adjusted automatically.

Uniform
The nodes along the edge are uniformly distributed.

Spacing
The spacing of the first node from the beginning of the edge (first cell height). When an edge is selected, an arrow appears along the edge. **Spacing 1** refers to the parameters at the beginning end of the arrow, and **Spacing 2** refers to the edge end where the arrow is pointing.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ratio
specifies the growth rate from one cell height to the next. Ratio 1 refers to the parameters at the beginning end of the arrow, and Ratio 2 refers to the edge end where the arrow is pointing, as shown in the figure below.

Max Space
specifies the maximum element spacing of the curve.

Curve direction
displays the curve direction with a yellow arrow along the curve at the midpoint. The arrow points from side1 (spacing 1 and ratio 1) toward side2 (spacing 2 and ratio 2). This is enabled by default.

Reverse direction
reverses the curve direction.

Note

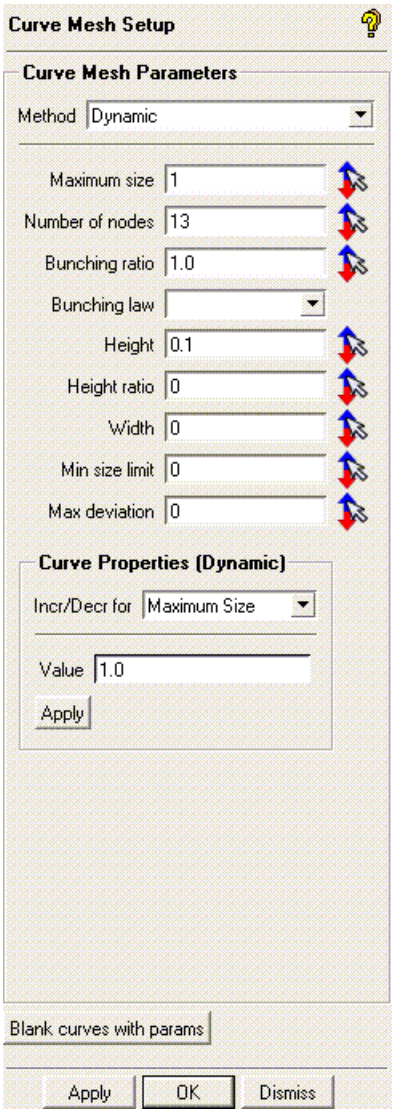
The **Reverse direction** button just flips side 1 and 2, it does not reverse the spacing, the bunching law, etc.

Adjust attached curves
adjusts the mesh sizes on attached curves with the specified parameters. This applies to curves that are attached at an angle between 60 and 120 degrees only. In order to select this option, the **Max size** parameter and the bunching law of the reference curve must be specified. The attached curves will automatically be assigned one of the following bunching laws, depending on the curve direction: Geometric 1, Geometric 2, or BiGeometric. The curve bunching of the attached curves will correspond to the values of **Height** and **Height Ratio** for the reference curve.

Remesh attached surfaces
allows you to change the mesh by changing the element size or number of elements on the curve. This option should be used when meshing has been completed, but you want to change the curve mesh sizes. The new surface mesh will automatically be generated.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 298: Curve Mesh Setup – Dynamic



Dynamic

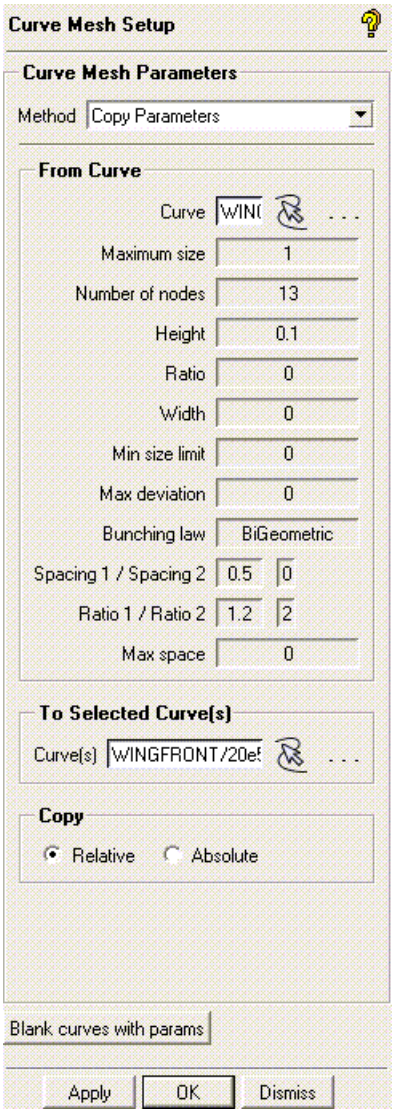
Select the curve parameter to be modified by clicking the increment/decrement icon ( ) next to it. This will display the values for that parameter for each curve. You can then select the value for a specific curve and use the left mouse button to increase the value or the right mouse button to decrease the value in increments set in the **Value** field.

Curve Properties (Dynamic)

allows you to specify the increment for specific curve mesh parameters in dynamic mode. Select the curve mesh parameter in the **Incr/Decr for** field, enter the value by which to increment/decrement the parameter in the **Value** field, and click **Apply** in the **Curve Properties (Dynamic)** group box.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 299: Curve Mesh Setup – Copy Parameters



Copy Parameters

allows you to copy curve mesh parameters to the selected curve(s).

From Curve

allows you to select the curve to copy the parameters from.

To Selected Curve(s)

specifies the curve(s) to which the parameters will be copied.

Copy

Relative

allows you to copy the parameters from the source curve relative to the curve length of the specified curve(s).

Absolute

allows you to copy the exact parameters from the source curve to the specified curve(s), regardless of curve length.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blank curves with params
blanks curves with prescribed parameters. Curves without parameters will remain visible.

Note

The number of nodes and the meshing laws specified take precedence in determining the number of nodes. Then spacings 1 and 2 are equally balanced, followed by ratios 1 and 2, which are also equally balanced, and finally **Max space** is considered.

Create Mesh Density

The **Create Mesh Density** option allows you to create or manipulate a density region. This is a polyhedral (or polygonal for 2D) zone in which one can prescribe a local maximum element size. This is useful for refining the mesh in a volumetric zone that is not adjacent to any geometry, for example, in the wake region of a vehicle. You can have density regions within one another, or partially intersecting one another.

Figure 300: Create Density DEZ

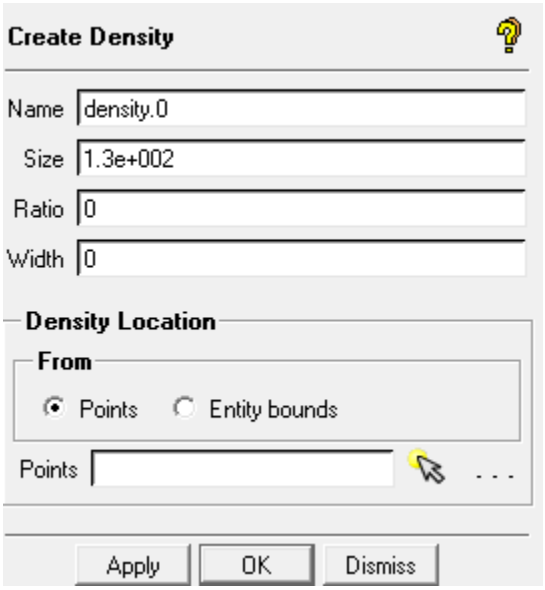
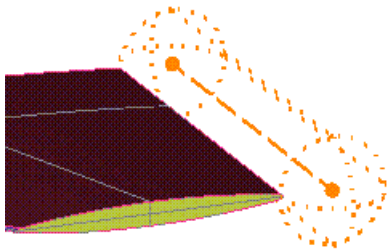
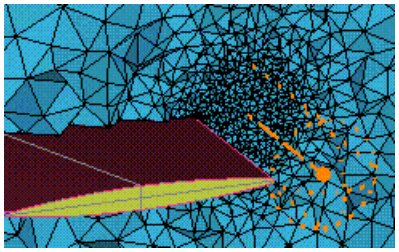


Figure 301: Example of Mesh Density



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A smaller mesh size within one density region will supersede that of a greater mesh size if they overlap or intersect. The density region does not have to be totally within the volumetric domain, but can intersect geometry and partially be within the dead zone.

Note

The density region is not part of the geometry, and mesh nodes are not constrained to the density region.

Note

This affects only Tetra, Cartesian, and Patch Independent surface mesh methods.

Name
specifies the name for the density region. Enter an appropriate name or accept the default.

Size
specifies the local maximum mesh size that can occur within the density region. This will be multiplied by the **Global Scale Factor**.

Ratio
specifies the tetra growth ratio away from the density region.

Width
For a density region, this specifies the number of layers (N) of the specified element size away from the boundary of the density region that should have a constant expansion ratio. The layer N + 1 will have a tetra size of the **Size** value multiplied by the **Ratio**.

For line and point densities, the **Size** value multiplied by the **Width** is the radius of the region that the density region influences.

Density Location
The location of the density region can be defined in the following ways:

- **From Points**

  Select one or more points to define the boundaries of the density region.

- **From Entity Bounds**

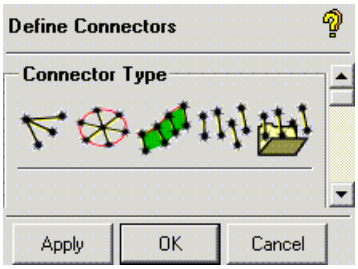
  Select an entity to define a density region within its boundaries.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define Connectors

The **Define Connectors** option allows you to define connectors. Connectors are meshing directives assigned to geometry so that when the geometry is meshed, mesh links are created between different parts. Line or shell elements are created to link parts with node to node connections. In general, mesh connectors are defined with the geometry so that if a geometry is updated parametrically, the connectors can be updated from one tetin file to a second tetin file. See **Main Menu > File Menu > Import Geometry > Reference Geometry** for more information.

Figure 302: Define Connectors Options



The various types of connectors are shown in Figure 302: Define Connectors Options (p. 329).

Arbitrary Connectors
Bolt Weld Connectors
Seam Weld Connectors
Spot Weld Connectors
Spot Weld From File

Arbitrary Connectors

The **Arbitrary Connectors** option allows you to create arbitrary connectors between any two entities. Each node of the start entity will have a connector to the end entity, but the end entity does not have to have all of its nodes connected. Arbitrary connectors are always created from the nearest point of the first entity to the nearest point of the second entity. The following options are available:

Arbitrary connector name
You can enter a name for the arbitrary connector, or use the default name.

Source
You can either select entities (generally curves) or existing parts as the source.

Target
Target entities are surfaces or parts containing surfaces. If only one or some of the surfaces of the part are to be selected for the target, then use the **Entities** method. If all the surfaces in the part are required to be selected, then use the **Existing Part** method.

Note

Ensure that the source and the target entities are selected in the correct order, select the source entity first and then the target entity.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Connector Part Name
The part name for the connectors is automatically generated, starting with ARB_WELD0.

Max projection
A value slightly more than the distance between the source and the farthest target surface.

Active
If disabled, the connector will not be created when remeshing.

Bolt Weld Connectors

The **Bolt Weld Connectors** option allows you to create bar elements inside a hole which all connect to one node at the center. These are also named bolt spiders because of their appearance. It is important to put the bar elements which circle the hole into a part of their own, because connectors will be created from all the bars in the specified part. If hole curves are put into a separate part before surface meshing, hole bars will automatically be in a separate part from the rest. This is because when the surface mesher runs, bar elements are put in the same part as the curves they lie on. If the part is changed after surface meshing, the bar elements must also be changed in order for the bolt weld connectors to be created. The following options are available:

Bolt hole name
A new part name for the bolt curves is automatically generated, starting with BOLT_CURVES0.

Bolt curves
specifies the curves representing the bolt hole.

Connector Part Name
specifies the part name for the connectors which is automatically generated, starting with BOLT_WELD0.

Num quad rings
is the number of quad layers to mesh around the bolt curves. This is subject to the space available around the curves.

Washer
Washers are represented by line (bar) elements and the thickness of the washer is represented by the quad layer(s) generated. The line elements will go from the inner and outer rings of the quad layer(s) to the center of the bolt hole to allow the load to get transferred from the bolt to both sides of the washer.

Note

To generate a washer at least one quad layer must be specified.

Active
if disabled, the connector will not be created when remeshing.

Seam Weld Connectors

The **Seam Weld Connectors** option allows you to create bar elements from all nodes that lie on the specified curve to a specified surface. The created connectors will be normal to the surface, so the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface elements may be automatically split in order to create nodes at the proper locations to connect the bars. If nodes are aligned around the seam within a tolerance, then the elements will not be split.

Seam connector name

The part name is generated automatically, starting with SEAM_CURVES0

Weld type

- **Point to point**

  creates a row of bar elements between the nodes of the source curves and the nearest nodes of the target surface mesh.

- **Curtain tris/quads**

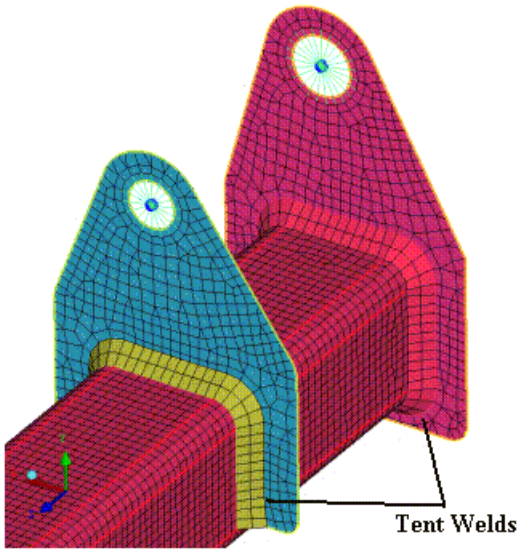
  creates quad elements with some tri elements in between the nodes of the source curves and the nodes of the target surface mesh. Curtain welds are generally intended to be defined on single edges to close a gap between two parts.

  Note

  In many cases, extending surfaces by using **Geometry > Create/Modify Surface > Extend Surface > Close gaps Between Midsurfaced Parts** is a more robust way of ensuring the creation of shell elements to represent the connection between parts.

- **Tent**

  is a specialized method that creates curtain elements with additional tent elements and defines particular meshing parameters on the source curves. The geometry is modified by surface extension or curtain surface functions. The Tent welds extend the surfaces to intersection and then diagonal quads represent the fillet welds.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tent Weld type
select either T welds or Lap welds.

Diagonals
For T welds, diagonals can be added to the **Positive** or **Negative** directions, or both. For Lap welds, diagonals can be added only to either the **Positive** or **Negative** directions.

No. layers
number of quad rows along the welds.

Tent weld spacing

– **Distance to Weld Toe**

The weld spacing can be determined by specifying the **Distance to weld toe**, the **Number of layers to toe**, and the **Growth ratio after weld** parameters.

– **Weld fillet radius**

The weld spacing can also be determined by specifying the fillet radius of the weld.

Source curves
specifies the curves representing the edge of surface to be seam welded to the target surface.

Note

The source curves information may not be retained when you try to modify an existing seam weld connector.

Target part
specifies the surface part to which the connectors will attach. The connectors will be normal to the surface. Since new nodes are projected, the surface elements must lie on top of a surface, and they must be in the same part as the surface.

Max projection
is a value slightly more than the distance between the source entity and the farthest target surface.

Part name
The connector part name is automatically generated, starting with SEAM_WELD0.

Element splitting

- **Remesh area (Tri/Quad)**

  remeshes the area of the seam weld after creating the mesh and connector.

- **Terminate**

  terminates the split to keep the mesh count down. It doesn't give as many high aspect ratio quads, but it creates many tri elements.

- **Propagate**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

Note

This may result in new quad elements with poor aspect ratios.

Spot Weld Connectors

The **Spot Weld Connectors** option allows you to create bar elements from all nodes that lie on the specified part to a specified surface. The created connectors will be normal to the surface, so the surface elements may be automatically split in order to create nodes at the proper locations to connect the bars. If the nodes are aligned within a tolerance, the surface elements will not be split.

Spot welds support 3T welds, which are 3 plates being welded together. For a 3T spot weld, pick 2 parts as Target parts.

A spot weld is intended to be used in the middle of a surface. It can also be defined on a surface boundary, with the condition that the source points should split the curve at the surface boundary.

The following options are available for Spot Weld connectors:

Spot Weld name
The part name is automatically generated, starting with SPOT_POINTS0.

Source points
specify points which represent one end of the connectors.

Note

The source point must be embedded in the source surface and both must belong to the same part.

Target parts
specifies the two parts that are to be connected.

Connector Part name
The part name for the connectors is automatically generated, starting with SPOT_WELD0.

Max projection
is a value is slightly greater than the maximum distance between two target parts.

Weld options

- **Point to Point**

  creates a line element attached to the nodes of both surfaces, and normal to both surfaces.

  – **Element Splitting**

  → **Remesh area (Tri/Quad)**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

remeshes the area of the spot weld after the mesh is created.

→ **Terminate**

terminates the split to keep the mesh count down and doesn't give as many high aspect ratio quads, but it creates many more tri elements

→ **Propagate**

propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

- **Mesh independent line**

This is like the point weld, but the nodes on the two sides do not have to be forced into the mesh. Instead, any nodes in the nuggets area are connected to the line element through a second set of line elements for both sets of surface mesh. Typically, the first line element is defined as an RBE2 element in Nastran, and the second set of line elements on each side are defined as RBE3 elements, which have a weighting factor based on how close it sits to the node of the line element. If the leg of the RBE3 is approximately zero, it gets a weighting value of one, and if it is large the weighting value is approximately zero.

– **RBE3 Part Name**

For a mesh independent weld, there is a weld element (that does not connect to the mesh), and connector elements that connect the weld element to the mesh. The connector elements to into the RBE3 Part. Typically, this weld is used for Nastran, where the line elements in this part should get an RBE3 Element Property.

- **Mesh independent hexa**

This is like the line independent weld, but the base line element (RBE2) is replaced by a Hexa element with a thickness defined by the diameter. Each of the four nodes at each surface has a RBE3 connection from its node to all nodes of the nearest element. The Hexa elements typically get a rigid material property, and the line element connectors are weighted as in a line mesh independent weld.

– **Weld Radius**

defines the weld diameter. The weld diameter (Radius * 2) would be equal to the diagonal of the Hex element for each face that is near the shells.

– **RBE3 Part Name**

Contains the line elements that connect the Hexa element to the mesh. These line elements should get an RBE3 element property.

- **Area weld**

An hourglass type of weld, where there is a nugget area or diameter, and a node in the center between the two surfaces meshes, where the center node is attached to any nodes within the diameter on either side. This is mesh independent, and the weld/line elements adapt to where the mesh is.

– **Weld Radius**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The weld connects all the nodes on the defined parts within the specified radius to the central node.

Active
if disabled, the connector will not be created when remeshing.

Spot Weld From File

The **Spot Weld from File** option allows you to create bar elements from the weld data file.

Spot Weld data file
enter or browse for the path to the Weld data file. See below for sample lines of a Weld data file.

Description	No.	X	Y	Z	PID1	PID2	PID3	Diameter
MASTIC	1	3046.278	531.872	1520.413	54325	54525		5
MASTIC	2	3095.952	528.354	1526.115	54325	54525		5
MASTIC	3	3145.952	525.258	1531.003	54325	54525		5
WSPOT	44	3340.525	695.365	537.187	54325	54525		
WSPOT	45	3370.485	695.982	524.562	54325	54525	54625	
MASTIC	46	3456.33	520.374	1543.035	54325	54525	54625	10
MASTIC	47	3260.055	525.304	1537.365	54325	54525	54625	10
MASTIC	48	3491.99	519.932	1543.285	54325	54525		10

Note

The first column must define the connection type. The X, Y, and Z columns define the coordinates of the connection location. The PID1, PID2, and PID3 columns define the parts connected. The Diameter column is important for MASTIC connections. For WSPOT connections, the Diameter column should be left blank.

Spot Weld report file
After welding, a Weld report file of this name will be generated and placed in the working directory. This will list all the welds, which ones failed and why. Common reasons for failure include missing or misnamed parts or badly placed welds.

Element splitting

- **Remesh area (Tri/Quad)**

  remeshes the area of the weld after the mesh is created.

- **Terminate**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

terminates the split to keep the mesh count down and doesn't give as many high aspect ratio quads, but it creates many more tri elements.

- **Propagate**

  propagates the split through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

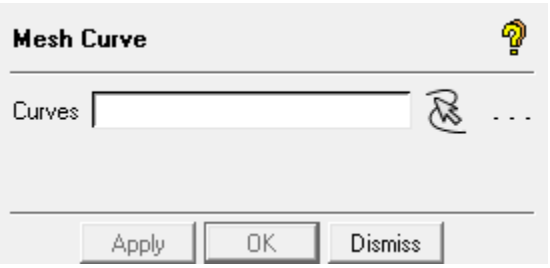
Connector Part name
The part name for the connectors is automatically generated, starting with SPOTWELDFILE0.

Max projection
is the value slightly more than the maximum distance between two target parts.

Mesh Curve

The **Mesh Curve** feature extracts 1D line elements from the selected curves. If the curve mesh size has been defined, it will be respected.

Figure 303: Mesh Curve DEZ



Compute Mesh

The **Compute Mesh** option allows you to generate the mesh specified by the mesher and various parameters.

Figure 304: Compute Mesh Options

The settings in the **Global Mesh Setup** options will be applied unless otherwise specified.
Compute Surface Mesh
Compute Volume Mesh
Compute Prism Mesh

Compute Surface Mesh

The **Compute Surface Mesh** option generates a surface mesh. By default the software tries to apply good meshing parameters for use in surface meshing, but you can apply additional controls by

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

changing the meshing parameters. The parameters that control the surface meshing are defined under Global Mesh Setup, Part Mesh Setup, and Surface Mesh Setup or Curve Mesh Setup

Overwrite Surface Preset/Default Mesh Type
if enabled, the specified mesh type will be used instead of the mesh type set in Global Mesh Setup > Shell Meshing Parameters.

Overwrite Surface Preset/Default Mesh Method
if enabled, the specified mesh method will be used instead of the mesh method set in Global Mesh Setup > Shell Meshing Parameters.

Input
specifies the geometry that will be used as input for the surface mesh.

All
meshes the entire geometry.

Visible
meshes the visible geometry.

Part by Part
meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From Screen
allows you to select the entities to be meshed from the display.

Note

For Patch Dependent meshing, if curves that form a closed loop are selected and meshed with one element type, and then remeshed using another element type, the original elements will not be deleted.

Compute Volume Mesh

The **Compute Volume Mesh** option generates a volume mesh using the selected volume mesh type and method. The descriptions of the different options and their parameters are described in Global Mesh Setup > Volume Meshing Parameters. The volume mesh will be applied using the parameters that are set there.

Load mesh after completion
This option applies to all the Compute Volume Mesh options. If disabled, the surface mesh will not be loaded into the GUI. This may be useful for big models.

The available volume mesh types are as follows.
Tetra/Mixed Mesh Type
Hexa-Dominant Mesh Type
Cartesian Mesh Type

Tetra/Mixed Mesh Type

There are three different Mesh Methods available for Tetra/Mixed Meshing: **Robust (Octree)**, **Quick (Delaunay)**, and **Smooth (Advancing Front)**. The different options for each mesh method are:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Robust (Octree) Mesh Method

This algorithm ensures refinement of the mesh where necessary (based on entity sizes and the curvature and proximity based refinement settings), but maintains larger elements where possible, according to the ICEM CFD octree algorithm and applied settings (such as tetra ratio). For additional information see The Octree Mesh Method.

Create Prism Layers

generates prism inflation layers into the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters, Part Mesh Setup, Surface Mesh Setup, and/or Curve Mesh Setup.

Create Hexa-Core

generates a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

Input

specifies the geometry that will be used as input for the Volume Mesh.

All

meshes the entire geometry.

Visible

meshes the visible geometry.

Part by Part

meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File

runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Use Existing Mesh Parts

forces the Octree Tetra mesher to align to the surface mesh of the selected existing mesh parts when the **All** or **Visible** option is selected for the input geometry. The Octree tetra mesh will be generated in its normal top down (volume first) method and then be "made conformal" with the existing surface mesh.

Quick (Delaunay) Mesh Method

uses the Delaunay Tetra mesher and a bottom-up meshing approach to generate a mesh. The surface mesh can be an existing mesh or will be created with the parameters defined under Global Mesh Setup > Shell Meshing Parameters or Surface Mesh Setup. The volume mesh will then be generated from this surface mesh. The Delaunay method is robust and fast.

Note

This method works with quads or tris or a combination of both. Quads will use pyramids to transition to the Tetras. The surface mesh can have multiple volumes and have multiple edge elements. However, the Delaunay mesh method requires a closed surface mesh, and cannot tolerate single edges, overlapping elements or duplicate elements. You can run a mesh check (**Edit Mesh > Check Mesh**) before generating the volume mesh. Also, sudden changes in element size, either adjacent or across a narrow volume gap, can cause quality issues or failure.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Prism Layers
generates prism inflation layers into the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters, Part Mesh Setup, Surface Mesh Setup, and/or Curve Mesh Setup.

Create Hexa-Core
generates a hexa-core mesh using a bottom-up meshing approach. It will retain the tri surface or prism mesh, delete the existing tetra mesh, and remesh the volume interior with Cartesian meshing. The tetra elements will be mapped to the tri or prism faces with the Delaunay algorithm.

The Hexa-Core mesh parameters are specified under Global Mesh Setup > Volume Meshing Parameters > Cartesian Mesh Type > Hexa-Core Mesh Method.

Volume Part Name
allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For the Delaunay method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, it is recommended that you enable the **Flood fill after completion** option (available in the Quick (Delaunay) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Note

When the **inherited** option is selected, the mesher will run the flood fill operation, even if **Flood fill after completion** is disabled under **Volume Meshing Parameters** in the **Global Mesh Setup**.

Input
specifies the geometry that will be used as input for the Volume Mesh.

All Geometry
meshes the entire geometry.

Existing Mesh
allows the mesher to align to the existing mesh.

Part by Part
meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File
runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

Smooth (Advancing Front) Mesh Method
This option will use the Advancing Front Tetra mesher to generate a mesh using a bottom-up meshing approach. The surface mesh will be created with the parameters defined under Global Mesh Setup >

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shell Meshing Parameters or Surface Mesh Setup. The volume mesh will then be generated from this surface mesh.

Note

The surface mesh should be one enclosed volume with no single edges, multiple edges, non-manifold vertices, overlapping elements or duplicate elements. Sudden changes in element size, either adjacent to one another or across a narrow volume gap, can cause quality issues or even failure.

The surface mesh must be either tri or quad elements for the Advancing Front mesh method.

The primary advantage of the Advancing Front Method is the ability to generate a smoothly transitioning Tetra mesh, with a volume growth ratio controlled by the Expansion Factor under the Global Mesh Parameters.

Create Prism Layers
generates prism inflation layers into the tetra volume mesh that is created, according to the prism meshing parameters that are specified under Global Mesh Setup > Prism Meshing Parameters, Part Mesh Setup, Surface Mesh Setup, and/or Curve Mesh Setup.

Volume Part Name
allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

If you select **inherited**, the mesher will place the volume element into the same part as an existing material point. For this method, the existing material point name will be used, but without its specific location. This option works well for models with only one material point. For more advanced models, such as conjugate heat transfer models or models with sections of porous media, it is recommended that you enable the **Flood fill after completion** option (available in the Smooth (Advancing Front) mesh method options) to use each material point with its location to determine the volume mesh part names for each region.

Input
specifies the geometry that will be used as input for the Volume Mesh.

All Geometry
meshes the entire geometry.

Existing Mesh
allows the mesher to align to the existing mesh.

Part by Part
meshes the selected parts one by one. This provides non-conformal mesh between part interfaces.

From File
runs the mesher in batch mode from an existing tetin file. Enter the name of the Tetin file or browse the file manager.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hexa-Dominant Mesh Type

This option will generate a Hexa-Dominant mesh using a bottom-up meshing approach. The Hexa-Dominant mesher starts with surface quad dominant mesh and uses an Advancing Front scheme to fill as much of the volume as possible. For simple volumes, it can fill it completely. For more complicated volumes, it usually fills several layers in from the surface with hexa elements and then fills the middle with tetras and pyramids. Then a diagnostic is run, and if those central elements are poor, the inner volume will be meshed again with the Delaunay mesher. The surface mesh will be created with the parameters defined under Global Mesh Setup > Shell Meshing Parameters or Surface Mesh Setup. The volume mesh will then be generated from this surface mesh or the specified input.

Input
allows you to select from the following types of input to generate the mesh.

All Geometry
meshes the entire geometry.

Existing Mesh
allows the mesher to align to the existing mesh parts.

Volume Part Name
allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The created mesh will be assigned to this part name.

Cartesian Mesh Type

This option will generate a Cartesian mesh using a top-down meshing approach. The mesher works by continuously refining the initial grid in a binary fashion in each dimension, and eliminating the non-volume cells, up to the specified maximum refinement. The Cartesian mesher does not require an existing surface mesh and will ignore meshing parameters defined to control local surfaces. The mesh will be refined until the finest cell size does not exceed the **Max element** size specified in Global Mesh Setup or the Surface Mesh setup.
Body-Fitted Mesh Method

Body-Fitted Mesh Method

This option creates unstructured hexa mesh based on Cartesian mesh and fits it to the geometry. This works for both CAD and STL geometries. The mesher can handle “dirty” geometries as long as the **Max element** size is larger than the gap size.

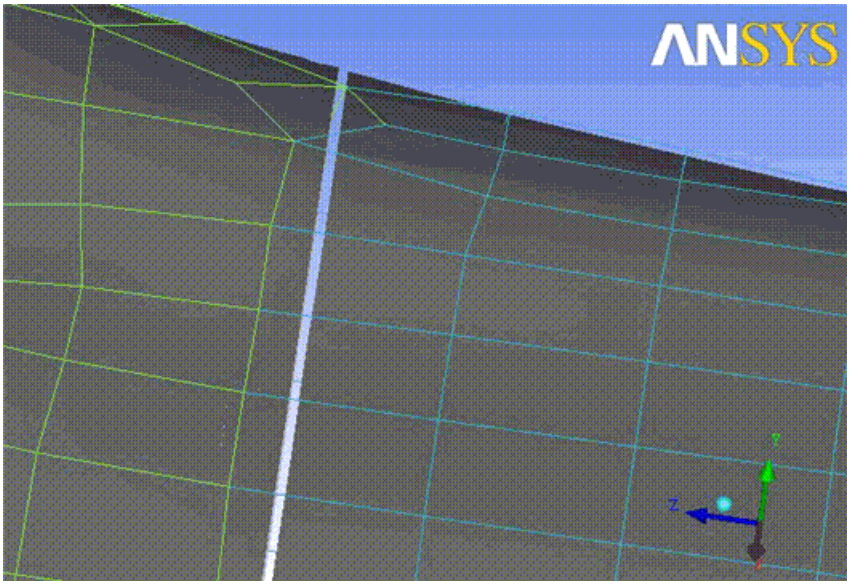
Note

The max element size should be smaller than the thickness of the model.

Figure 305: Handling of Gaps in the Geometry (p. 342) shows an example where the gap in the geometry is ignored by the body-fitted mesh method.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 305: Handling of Gaps in the Geometry



Volume Part Name

allows you to select from the list of existing volume parts, select a part from the screen, or supply a new name. The inherited option will use the material points in the geometry to determine the part name for each region. The created mesh will be assigned to this part name.

Enforce Split

These options allow you to use a pre-existing Cartesian mesh as the basis for the body-fitted Cartesian algorithm. This Cartesian mesh can be created elsewhere or can be created from ANSYS ICEM CFD Hexa Blocking. It can include biasing and various aspect ratios. It can also be aligned with various features of the geometry.

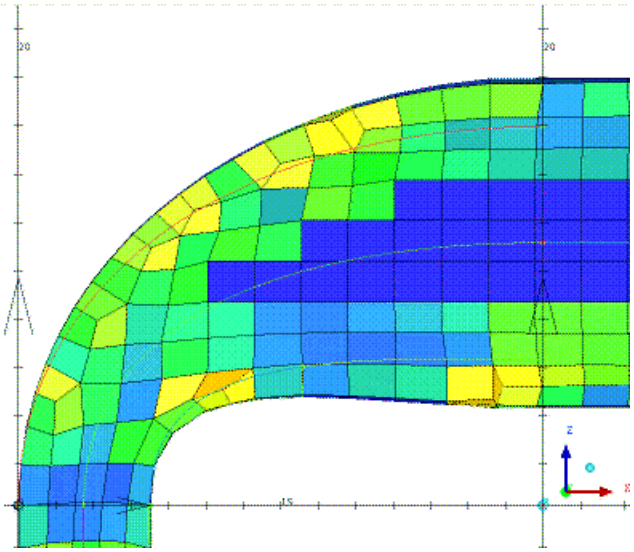
None

does not enforce a pre-existing Cartesian mesh, but rather creates a Cartesian mesh within the BFCart algorithm. This background mesh may have a given aspect ratio and may be aligned with the LCS. These options are controlled by the **Global Mesh Parameters** for BFCart.

Figure 306: BFCart Mesh Generated Using the None Option (p. 343) shows an example where this option has been used.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 306: BFCart Mesh Generated Using the None Option

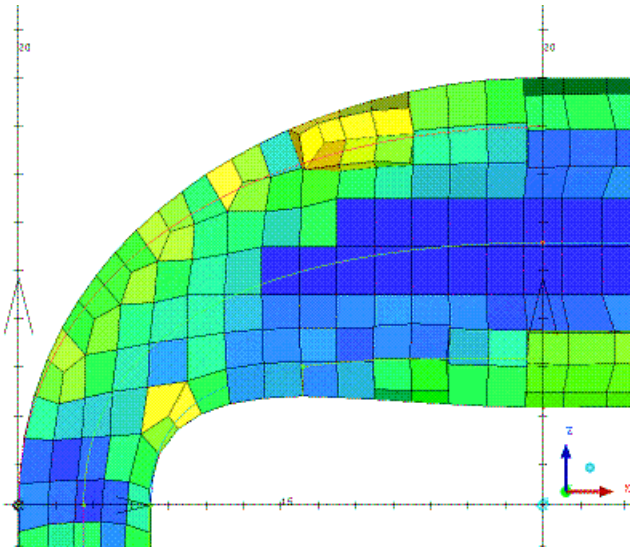


Initial

In order to body fit, the BFCart algorithm must take the inverse of the Cartesian mesh. The end result of the process is that the nodes of the original Cartesian mesh become the centers of the resulting mesh. The split lines of the original mesh end up between the split lines of the final mesh.

Figure 307: BFCart Mesh Generated Using the Initial Option (p. 343) shows an example where this option has been used.

Figure 307: BFCart Mesh Generated Using the Initial Option



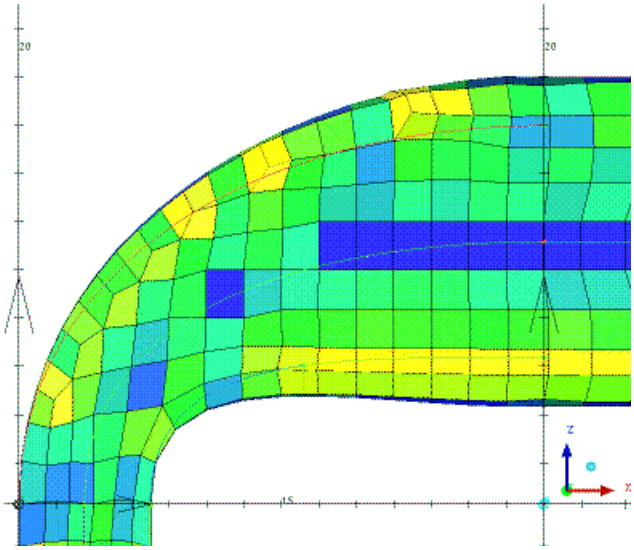
Final

With the **Final** option, the supplied Cartesian mesh is inverted before the rest of the process begins so that the later body fitting inversion changes it back and the final grid can line up with the original. Due to internal complexities, this only works if the “uniform” Cartesian mesh is used. It does not work for 2 to 1 hanging node Cartesian mesh.

Figure 308: BFCart Mesh Generated Using the Final Option (p. 344) shows an example where this option has been used.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 308: BFCart Mesh Generated Using the Final Option



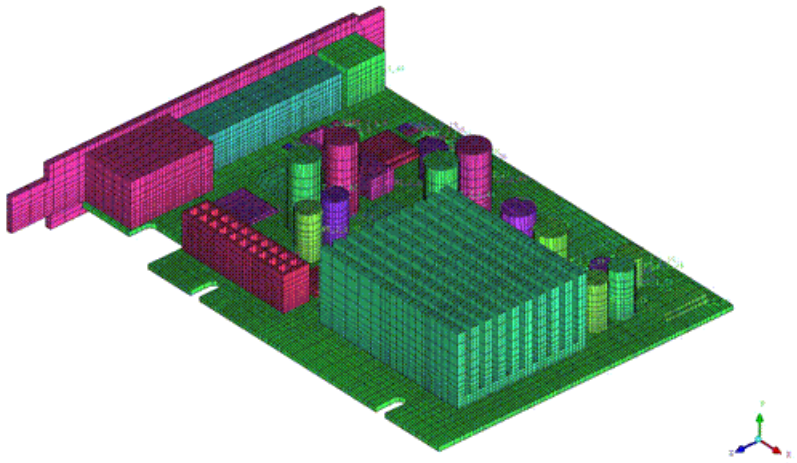
Cartesian file

specifies the Cartesian grid file to be used when starting from an existing Cartesian grid file rather than generating one based on the mesh parameters of the current model. When used in conjunction with the **File > Blocking > Write Cartesian Grid** option to export a block file as a Cartesian grid, you will be able to use block splits and edge parameters to control Cartesian bunching, distribution and aspect ratio. The imported mesh is used as the starting point before body fitting takes place.

Key-Points

The **Key-Points** option can be used to align the body-fitted Cartesian mesh with corner points. Behind the scenes, this uses Key-Point blocking to create a Cartesian file aligned with the points and then uses the **Final** option with that file to align the Cartesian splits with the points. This option is useful for models having features aligned in Cartesian directions. An example of a circuit board is shown in Figure 309: BFCart Mesh Generated Using Key-Points (p. 344) with several features aligned in the Cartesian directions.

Figure 309: BFCart Mesh Generated Using Key-Points



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tolerance
specifies the minimum distance between adjacent grid lines. The default value is 0.0. If the default value is used, the tolerance will be computed internally based on the minimum entity dimension.

Inflation
contains options for growing the body fitting inflation layer.

Defined
grows the inflation layer on the parts for which the **prism** option is enabled in the **Part Mesh Setup** dialog. The setting works as an on/off toggle. The other **Part Mesh Setup** options for prisms, including initial "height", "ratio" and number of layers ("width"), are not yet used for BFCart inflation.

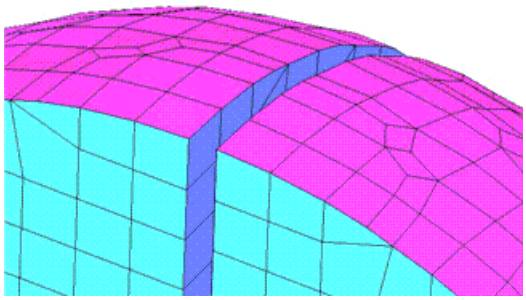
All
grows the inflation layer on all parts.

None
does not grow the inflation layer regardless of the settings in the **Part Mesh Setup** dialog.

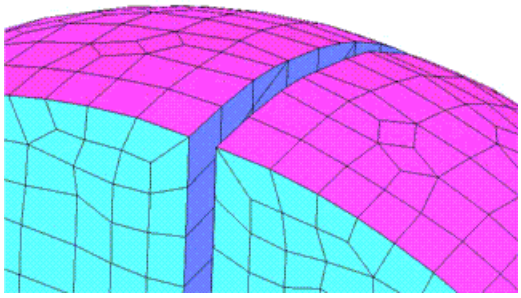
Figure 310: Selective Inflation for the Body-Fitted Cartesian Mesh (p. 345) shows the inflation options available for the Body-Fitted Cartesian mesh generated for a sphere with a gap.

Figure 310: Selective Inflation for the Body-Fitted Cartesian Mesh

(A) **Inflate Parts** set to **None** — The mesh quality is poor as the stair step mesh is projected to the round surfaces of the sphere.

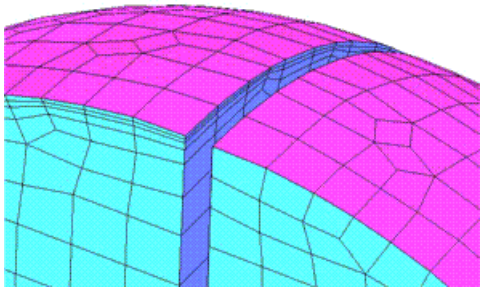


(B) **Inflate Parts** set to **All** — All parts are inflated, including the flat surface in the gap. The mesh quality is not as good as it could be due to miter elements generated in some corners.



(C) **Inflate Parts** set to **Defined** — Only the sphere surfaces are inflated, while the gap surface part is not inflated. The inflation from the curved walls runs into the gap and gives the best resulting quality. On the left of the gap, the inflated hexas are split propagated to give better boundary resolution.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Compute Prism Mesh

Generates prism inflation layers into the volume mesh to resolve boundary layer effects efficiently. Existing tetra mesh is restructured to maintain proper connectivity and quality. The prism layers are typically orthonormal to the viscous wall boundaries. The prism meshing parameters can be specified under Global Mesh Setup > Prism Meshing Parameters, Part Mesh Setup, Surface Mesh Setup, and/or Curve Mesh Setup.

Input

allows you to select the mesh in which the prism inflation layers will be generated.

- **Existing Mesh**

  creates prism mesh in the existing mesh.

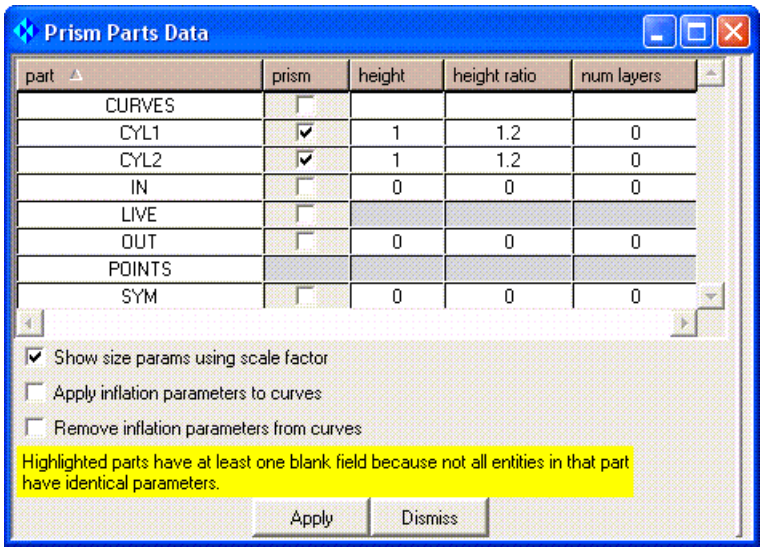
- **From File**

  runs the Prism Mesher in batch mode from an existing mesh (*.uns) file. Enter the name of the file or browse the file manager.

Select Parts for Prism Layers

allows you to define the parameters of prism layers for different parts.

Figure 311: Select Parts for Prism Layer



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the parts that prism layers will be applied to. For each part, the Height, Height Ratio and the Num Layers can be specified. Volume, Surface and/or Curve parts can be selected. When using the Prism mesher in ANSYS ICEM CFD, the default prism parameters are set under Mesh > Global Mesh Setup > Global Prism Settings. However, parameters that are entered here at the Part level will override the Global settings. In addition, the entity settings of Height, Height Ratio, and Number of Layers will override the Part level settings. Entering zero for any of the settings will default back to the Global Settings.

If there are multiple volume materials, and none have the **prism** option enabled, then prism mesh will grow from the selected surface parts into the adjacent volumes. If only certain volume parts have **prism** enabled, then prism mesh will be grown only into those volume parts. After you enter the parameters, click **Apply**, and then **Dismiss** to close the window.

For 3D, the prisms are grown (inflated) from the shell (tri or quad) elements of each part. This can be done with or without a volume mesh, but having a tetra volume mesh during inflation helps with collision avoidance and ensures that you will have a volume mesh after prism generation is complete. For 2D, the prisms are grown (inflated) from the curve parts into the selected surface parts (must be selected). The 2D prism only works if Advanced Prism Meshing Parameters > Blayer 2D is enabled.

Note

- You can compute a prism mesh without an input surface model loaded. Prism will generate a temporary faceted surface model from the input mesh.
- Different prism heights can be specified on adjacent parts, though a transition region with unspecified height is required in between these parts.
- If a surface part separates two or more volume parts, select the volume parts on the side of the surface you want to grow the prisms. If you select both sides, prisms will grow in both directions from the surface part. If you want different prism properties on either side of a surface grown into 2 volumes, do one at a time (run prism iteratively).
- If adjacent tri-element parts have heights that differ by more than a factor of 2, the prism mesher may fail (this limit is controlled in the **Advanced Prism Meshing Parameters**). Not setting a height for Prism, (here or in the **Global Prism Parameters** DEZ) will allow the height to float. You can also allow the height to float globally and set specific initial heights per part or on an entity by entity basis.

Load mesh after completion

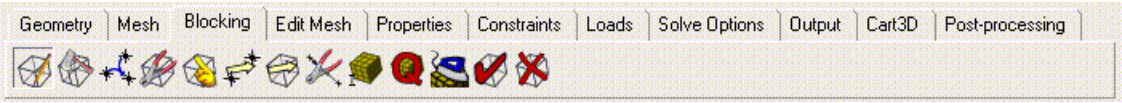
if disabled, the mesh file will not be loaded into the GUI. This may be useful for big models.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Blocking

Figure 312: Blocking Menu



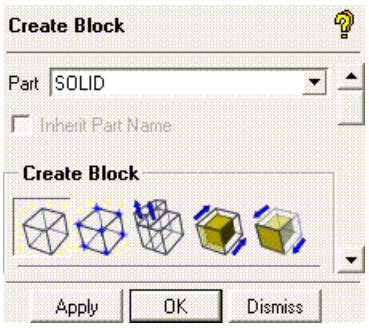
The **Blocking** tab contains the following options to create blocking over any geometry.

Create Block
Split Block
Merge Vertices
Edit Block
Associate
Move Vertex
Transform Blocks
Edit Edge
Pre-Mesh Params
Pre-Mesh Quality
Pre-Mesh Smooth
Block Checks
Delete Block

Create Block

The **Create Block** option establishes new blocking.

Figure 313: Create Block Options



The following options are available for creating blocks:

Initialize Block
From Vertices/Faces
Extrude Face
2D to 3D Blocks

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D to 2D

Note

If a blocking file is not loaded, then only the first option, **Initialize Blocks**, will be active. If a blocking file is already loaded, then all the options will be active.

Inherit Part Name

when enabled, the created blocks will inherit their part name from the entities they were created from. When initializing 2D surface blocking, the part name can be inherited from underlying surface parts. For other blocking operations, such as Extrude Face or 2D to 3D), which start from an existing blocking, the part name of the new blocks is inherited from the blocks they were created from. If this option is disabled, all blocks will be in the part name specified in the drop-down list at the top of the **Create Block** DEZ.

Note

Mesh created within each block will belong to the same part as its block. This part name is also used to tag the elements on export and is useful for assigning material (or zone) properties in the solver.

Initialize Block

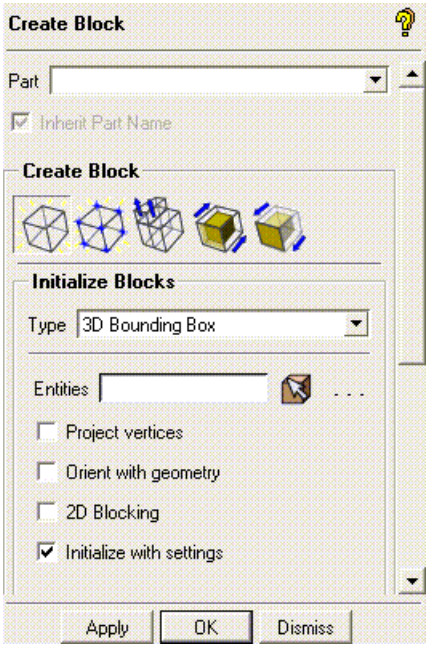
The **Initialize Block** option allows you to initialize blocks with the following options:

3D Bounding Box

allows you to create a 3D block enclosing the selected entities. If no entities are selected, the block will encompass the entire geometry.

ANSYS ICEM CFD Hexa blocking is typically done in a top down method that starts from a single large block which is subdivided down to create the topology. Initializing this 3D bounding box is the first step.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note

Not selecting any entities is more efficient, particularly for scripting, if you intend to block the geometry extents.

Project vertices
when enabled, the initial block vertices will be moved to the nearest locations on the geometry.

Orient with geometry
attempts to find the best fit of the geometry in any orientation, and to create the smallest block possible around the geometry selected.

2D Blocking
enables the creation of a surface blocking composed of six 2D face blocks forming a box around the geometry.

Initialize with settings
allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under **`.aienv_options`** for block initialization. For example, you can set the default bunching ratio to 1.2 and the multigrid level to 3 and save the settings. Future block initialization with the **Initialize with settings** option enabled will use these settings as default.

This option is enabled by default.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Surface Blocking

allows you to create automatic surface blocking for surface-only mesh. It creates a 2D block for each surface. When **Inherit Part Name** is enabled, the part name will be inherited from the underlying surface parts.

Note

This can be a precursor for the 2D to 3D Fill, i.e., Multi-zone method.

Note

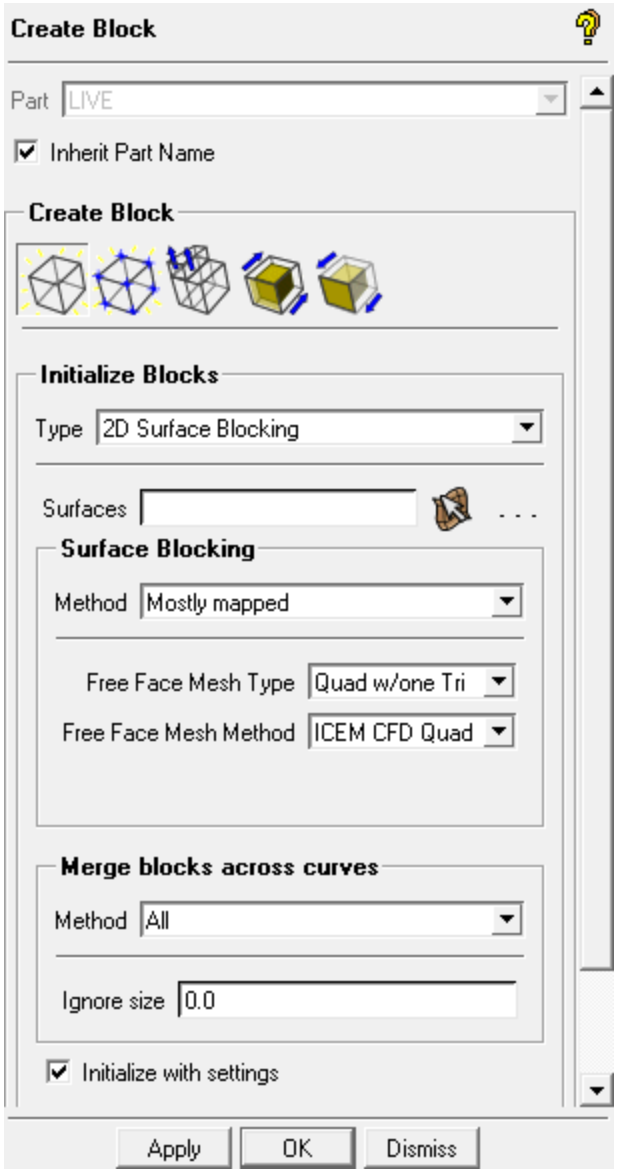
If surface mesh sizes (max size, height, and height ratio) have been previously set, they will be used to calculate the distributions for the blocking edges.

Note

Geometry topology information is required to establish connectivity between the blocks. You can check topology by right-clicking on Curves in the model tree and enabling the **Color by Count** option. Make sure that you have double edges (red curves) between surfaces that you want connected. Single edges (yellow curves) may indicate a gap between surfaces. **Build diagnostic topology** if necessary.

2D blocking can be created with the following options:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Surfaces

allows you to select specific surfaces for initializing 2D surface blocking. If none are selected, all surfaces will be initialized.

Surface Blocking

contains options for the surface blocking method, free face mesh type, and free face mesh method.

Method

specifies the surface blocking method. The following options are available:

Note

Blocks can be converted between free and mapped blocks using **Edit Block > Convert Block Type**.

Free

creates all free (unstructured) 2D blocks. Unstructured blocks can have any number of sides, and have a different number of nodes on opposite sides, with a resulting nonuniform element

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pattern. The mesh inside free blocks is paved (recursive loop algorithm) and can be all quad, quad dominant, quad with one tri, or all tri.

Note

The default mesh type that will be used is defined under **Settings > Meshing > Hexa/Mixed > Unstruct face mesh type**. You can override the default setting by changing the mesh type in the **Free Mesh Type** list. After the 2D Surface Blocking is generated, you can modify the mesh type for any particular Free block using the **Blocking > Edit Block > Convert Block Type** option.

Some mapped

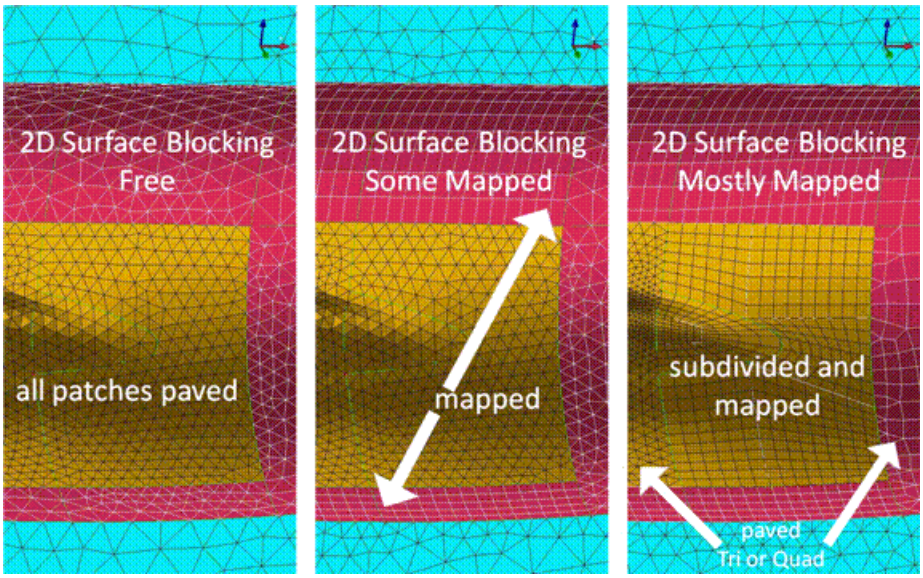
maps surface patches with 4 corners (possibly more sides). A mapped surface has matching numbers of nodes on opposite sides. The mesh is "mapped" across to the opposite side of the block and is always quad surfaced mesh. Remaining surfaces with more or less than 4 corners are blocked with "free" blocks as above.

Mostly mapped

tries to subdivide surface patches with more or less than 4 corners into mappable patches. For instance, a 3 cornered surface is divided into a quarter O-Grid (Y-Block) pattern. A half circle is divided into a half O-Grid (C-Block) pattern, other arbitrary shapes are simply divided. Other surface patches with 4 corners are also mapped. Any remaining patches or sections of patches which still cannot be mapped (more or less than 4 corners) are blocked with "free" blocks as above.

Figure 314: Surface Blocking Methods (p. 354) shows the **Free**, **Some mapped**, and **Mostly mapped** surface blocking methods.

Figure 314: Surface Blocking Methods



Swept

creates surface blocks in preparation for the **2D to 3D Fill > Swept** operation. Sides of sweepable bodies are mapped, sources are free or mapped and copied to the targets. This method can handle multiple sources and targets.

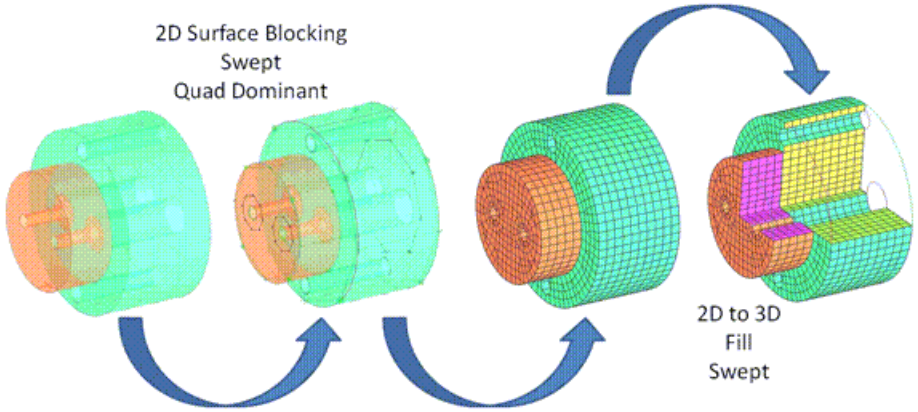
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Swept Surfaces

This option is available only for the Swept Method. Select all the source and target surfaces (multiple sources and targets are supported). If it is easier, select all surfaces involved (sources, targets, sides). If you intend to sweep everything, do not select surfaces at all and everything will be selected.

Figure 315: Surface Blocking-Swept (p. 355) shows the swept surface blocking. In this case, geometry topology was built between the two parts so that the sweep could pass through both parts.

Figure 315: Surface Blocking-Swept



Note

The 2D Surface blocking is automatically imprinted to deal with multiple source and target situations.

Note

As with all the 2D surface blocking methods, geometry connectivity translates into blocking connectivity, so make sure to check topology.

Free Face Mesh Type

specifies the type of mesh for the free blocks. The following options are available:

- All Tri
- All Quad
- Quad Dominant
- Quad w/one Tri

Free Face Mesh Method

specifies the process for meshing free faces. The following options are available:

- **ICEM CFD Quad** - method is based on a recursive loop-splitting algorithm.
- **Gambit Pave** - method is based on the Gambit paving advancing front algorithm.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Auto** - method lets the program determine the method based on size and curvature.

Merge blocks across curves

By default, surface patches are defined by each surface. However, surfaces can be combined to improve the blocking or prevent slivers. This is controlled or limited by dormant curves or the **Ignore size** tolerance.

Behind the scenes, a "loop" is formed around the perimeter of each surface. If two blocks are merged across a curve, the loops on either side are simply replaced by a loop that includes the perimeters of the 2 surfaces combined. The geometry (curves and surfaces) is unaffected by the operation.

Method

All
will merge a sliver surface with its larger neighbor if the characteristic edge length is less than the tolerance.

Respect non-dormant
will not merge if the curve between the surfaces is not dormant. This does not mean it will force a merge if the curve is dormant. Tolerance still determines the merge.

None
will not merge based on tolerance.

Merge dormant
will merge across dormant curves regardless of tolerance.

Note

To make a curve dormant, delete without the **Delete Permanently** option. The **Build Topology > Filter curves** option will also make curves dormant based on a feature angle. To restore a dormant curve, use the **Geometry > Restore dormant entities** option. To view dormant curves, right-click on Curves in the model tree and enable **Show Dormant**.

Note

After creating the 2D blocking, the **Edit Block > Merge Blocks** option can be used to achieve a similar result.

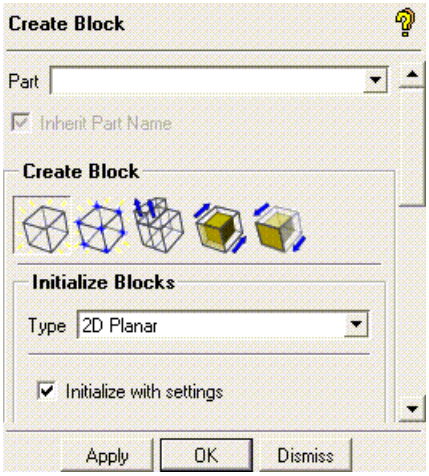
Ignore size
specifies the tolerance that **Merge blocks across curves** uses to determine whether to merge sliver blocks to the adjacent block. The sliver block's distance across the thin edge must be smaller than this value for the block to be merged to the adjacent block.

Initialize with settings
allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under `.aienv_options` for block initialization. For details, refer to Initialize with settings (p. 351) described above.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Planar

allows you to create a 2D Planar block in the XY plane fitting around the entire geometry. 2D planar blocking is intended to work in the XY plane, and moving it to another orientation will make it difficult to position internal vertices. It is better to rotate your geometry to the XY plane where Z=0.



Initialize with settings

allows you to use the user-defined settings set in the **Hexa/Mixed Meshing Options** DEZ (**Settings > Meshing Options > Hexa/Mixed**) and saved under **`.aienv_options`** for block initialization. For details, refer to Initialize with settings (p. 351) described above.

From Vertices/Faces

The **From Vertices/Faces** option allows you to create blocks from vertices or faces in either 2D or 3D. The available options are described in the following sections:

3D Blocks
2D Blocks

3D Blocks

The following 3D block types can be created:

Hexa

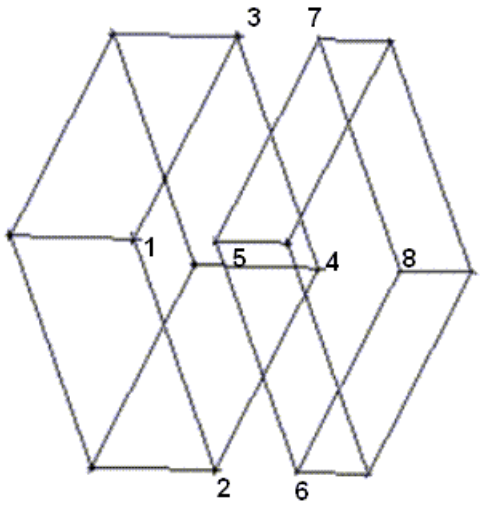
allows you to create 3D Hexa blocks by specifying the corner vertices or appropriate faces of adjacent blocks.

Corner Method

Select eight vertices (or locations) as shown in Figure 316: Selection of Vertices/Locations for the Corner Method (p. 358) to create 3D Hexa blocks.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 316: Selection of Vertices/Locations for the Corner Method



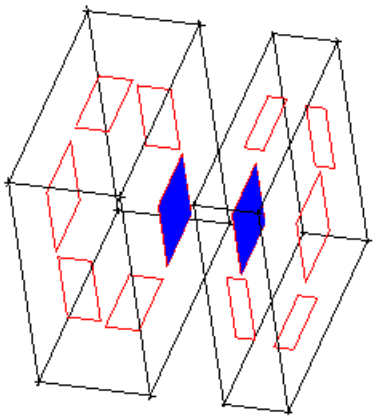
While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 8 vertices/locations.

Faces Method

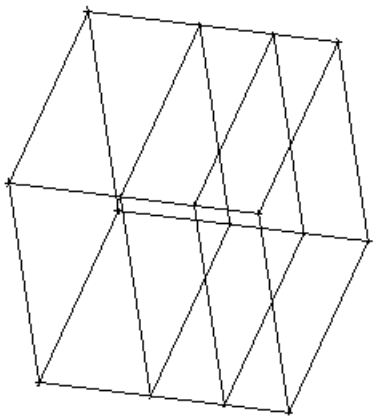
Select faces of adjacent blocks as shown in Figure 317: Selection of Faces for the Faces Method (p. 358) to create a new 3D Hexa block.

Figure 317: Selection of Faces for the Faces Method



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

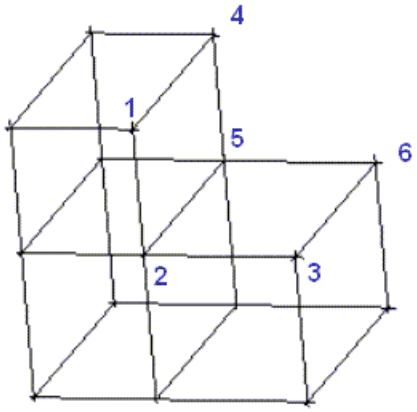
Figure 318: 3D Block Created



Quarter O-Grid

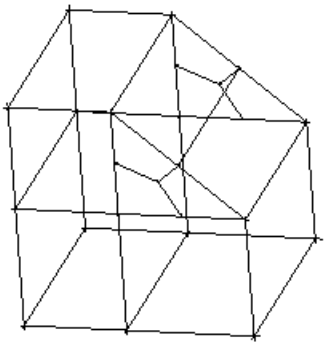
allows you to create the advanced topology known as a Y-Block or Quarter O-Grid. This topology is used to fit three Hexa Blocks into a wedge. Select six vertices/locations as shown in Figure 319: Selection of Vertices/Locations for Quarter O-Grid (p. 359) to create the Quarter O-Grid. The three vertices of one side of the wedge must be selected first, in clockwise or counter clockwise order. Then the remaining three vertices can be selected. It is important that the 4th vertex selected should be connected to the 1st, 5th to the 2nd and 6th to the 3rd respectively.

Figure 319: Selection of Vertices/Locations for Quarter O-Grid



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 320: Quarter O-Grid Created



While selecting a combination of vertices and locations:

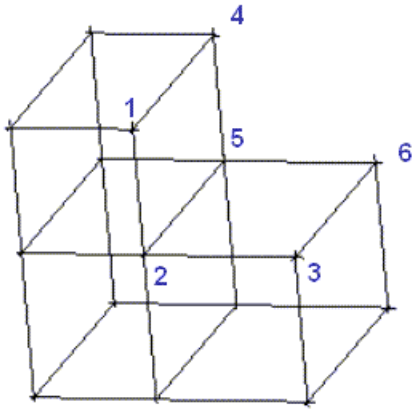
1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 6 vertices/locations.

Degenerate

allows you to create a degenerate block which is a prismatic block with 5 sides. Previously, these could only be created by collapsing one side of a hexa block to create a wedge. The bottom up creation follows the exact same method as the Quarter O-Grid (Y-Block) creation. However, only one block is created and there will be a row of prism elements along one edge. Many solvers can not handle this type of blocking, so be sure to consult your solver manual before using degenerate blocks.

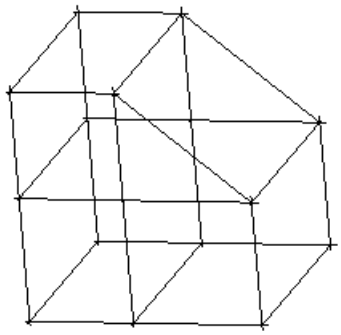
Select six vertices/locations as shown in Figure 321: Selection of Vertices/Locations for Degenerate Block (p. 360) to create the Degenerate block. The 4th vertex selected should correlate to the 1st, the 5th to the 2nd, and the 6th to the 3rd.

Figure 321: Selection of Vertices/Locations for Degenerate Block



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 322: Degenerate Block Created



While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 6 vertices/locations.

Swept

allows you to create a swept block. Select a face of a block. All the blocks in the direction perpendicular to this face will be made into swept blocks. A swept (free) block results in an unstructured mesh with some tri and prism elements.

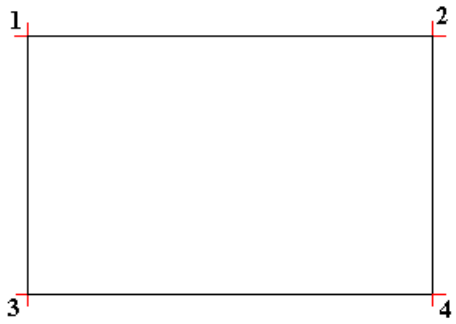
Note

This operation converts mapped blocks to swept blocks. If there is a swept or free block included in the blocks that are to be converted, there may be an error.

Sheet

allows you to create a structured sheet block from 4 vertices. The vertices should be selected in the order of a Z pattern (shown in Figure 323: Selecting Vertices/Locations for a Sheet Block (p. 361)). Click the middle mouse button to confirm the selection of the vertices.

Figure 323: Selecting Vertices/Locations for a Sheet Block



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Free-Sheet
allows you to create a free sheet block from 4 vertices.

2D Blocks

Note

The **2D Blocks** option allows you to create a block for a 2D blocking only. To create a 2D (sheet) block with a 3D blocking, even if the only 3D block is unstructured, you need to use the Sheet (p. 361) option described in the 3D Blocks (p. 357) section.

The following 2D block types can be created from vertices:

Mapped
allows you to create 2D blocks from any four specified vertices or locations.

In Figure 324: Selection of Vertices for 2D Mapped Block (p. 362), four vertices are selected to create a mapped block.

Figure 324: Selection of Vertices for 2D Mapped Block

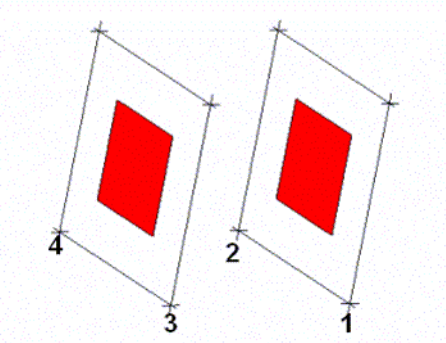
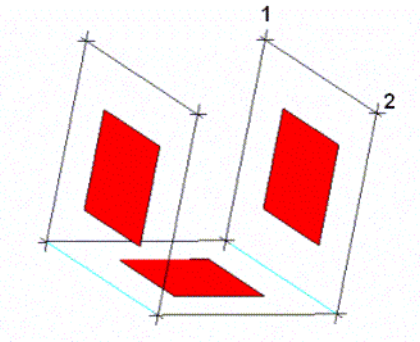


Figure 325: Mapped Block Created



While selecting a combination of vertices and locations:

1. First select the vertices to be used, ensuring that the order of selection is appropriate.
2. Click the middle-mouse button to confirm the selection of the vertices.
3. Select the remaining locations on the geometry (not necessarily points) to complete the selection of 4 vertices/locations.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In Figure 326: Selection of Vertices and Locations for 2D Mapped Block (p. 363), a mapped block is created from two vertices and two locations.

Figure 326: Selection of Vertices and Locations for 2D Mapped Block

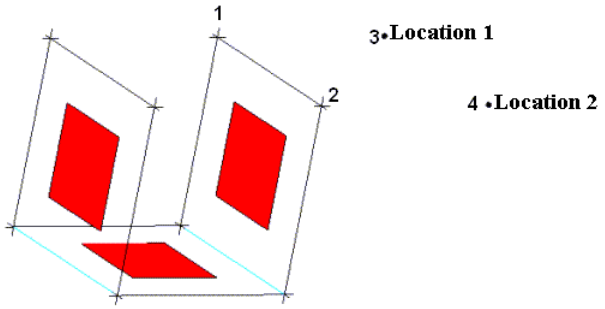
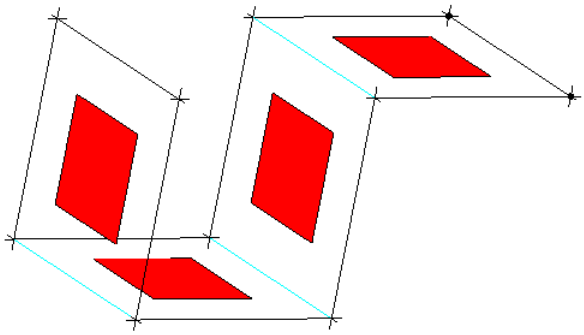


Figure 327: Mapped Block Created



Free

allows you to create a free block using the selected vertices.

Extrude Face

The **Extrude Face** option is available only for 3D blocking. The following methods can be used to extrude block faces.

Interactive
Fixed Distance
Extrude Along Curve

Interactive

Select the face(s) to be extruded with the left mouse button and then press the middle mouse button and drag to interactively extrude the face(s). When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same blocking material).

Fixed Distance

Select the face to be extruded and specify the distance for normal extrusion. When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

blocking material). The face selection and extrusion of the block is shown in Figure 328: Face Selected for Extrusion by Fixed Distance (p. 364) and Figure 329: Extrusion Completed (p. 364).

Figure 328: Face Selected for Extrusion by Fixed Distance

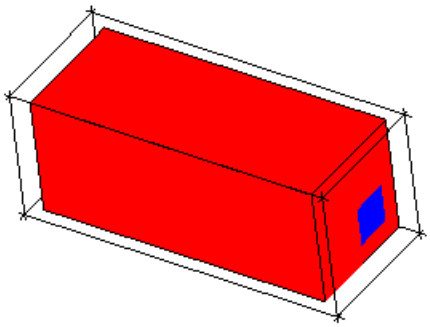
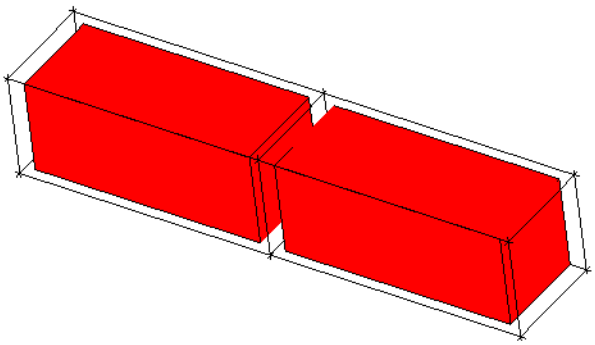


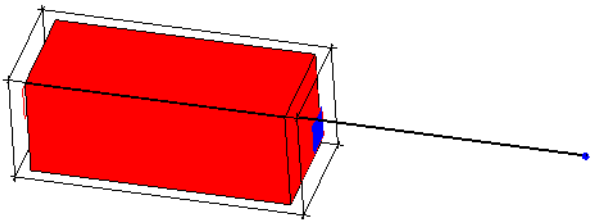
Figure 329: Extrusion Completed



Extrude Along Curve

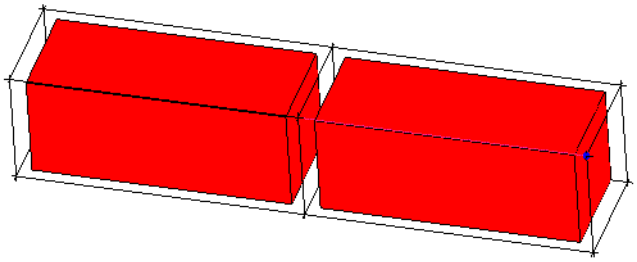
Select the face to be extruded, the curve along which the extrusion is to occur, and the end point of the curve. When **Inherit Part Name** is enabled, the new extruded block inherits the part of the adjacent volume block (is in the same blocking material). The face selection and extrusion of the block is shown in Figure 330: Face and Curve Selected for Extrusion Along Curve (p. 364) and Figure 331: Extrusion Completed (p. 365).

Figure 330: Face and Curve Selected for Extrusion Along Curve



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 331: Extrusion Completed



2D to 3D Blocks

The **2D to 3D** option allows you to convert 2D or surface blocking to 3D. The following options are available:

MultiZone Fill

The **MultiZone Fill** option converts surface blocking into 3D blocking. This requires a closed volume(s) of structured and/or unstructured surface blocks. It can produce a structured blocking if all the surface blocks are structured 4 sided blocks, otherwise unstructured or swept blocks may result.

Figure 332: 2D Blocking – Before Fill

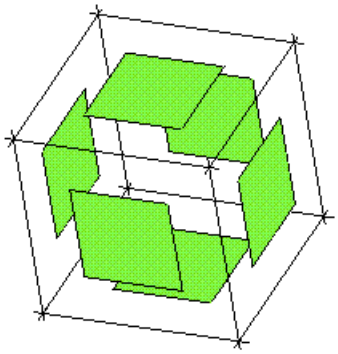
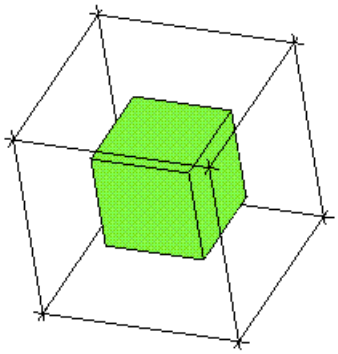
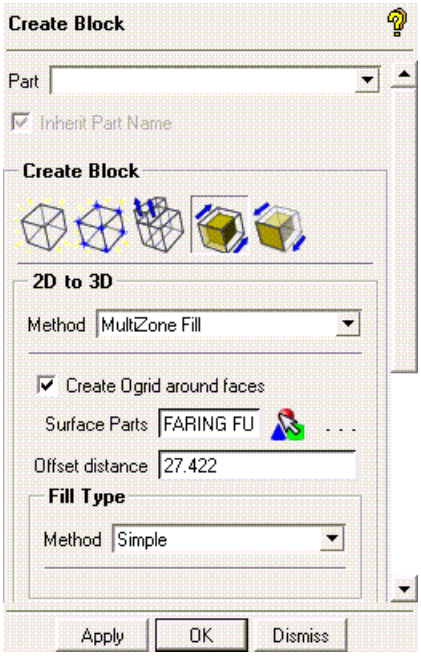


Figure 333: After Fill – 3D Blocking



The following options are available for the MultiZone Fill Method.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Create Ogrid around faces

allows you to add O-grid layers on the selected parts.

Note

When using this method for blocking a model with a floating internal volume, this method has difficulty connecting the internal and external volumes and the internal volume may merely be superimposed within the outer volume. Splitting the model with a connecting plane will allow a topological connection and solve this problem.

Surface Parts

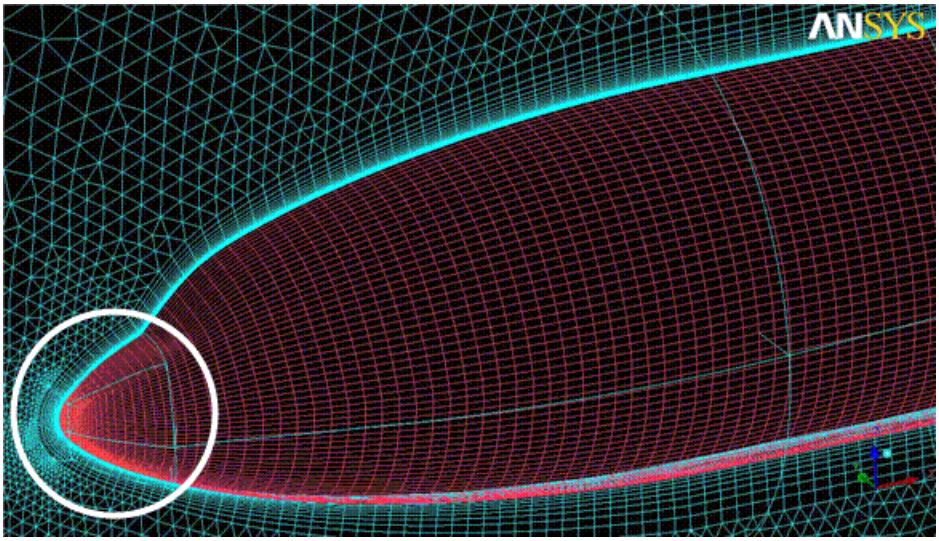
allows the selected parts to be marked for O-grid layer creation. These parts can be default selected by setting prism parameters in the **Part Mesh Setup** parameters menu. The other prism settings such as initial height, ratio and number of layers can also be set. You can then either accept the default selection here or add/remove parts as desired. The number of layers will be constant throughout a boundary layer, but varying the initial height will affect the total height above those surfaces.

Offset distance

specifies the height of the O-grid boundary layer calculated from the **Part Mesh Setup** parameters for height, ratio, and number of layers set for individual parts. If the **Part Mesh Setup** parameters are not set, the offset distance will be calculated from the **Global Prism Parameters**. The value displayed is the average height calculated, however, the O-grid boundary layer will be created based on the offset calculated for each individual part. For example, in Figure 334: Variable O-grid Height (p. 367), the initial height on the highlighted surface was set to half the height of the other surfaces.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 334: Variable O-grid Height



If you override this calculated height, it will apply to the entire O-grid. It will prioritize the initial height and ratio and adjust the number of layers to fit the user-specified total height.

Note

The number of layers must be constant for the entire O-grid layer, if you set varying numbers of layers, the greatest will be used.

Note

The variable O-grid height affects the initial O-grid generation to make the process more automatic but Multi-zone builds an editable blocking with flexible vertices. So, any edge length, including O-grid heights, can easily be changed interactively.

Fill Type

Simple
creates a simple free block or volume region. This saves time on complex models by not attempting to decompose the volume into mapped or swept blocks. This is recommended for external aerodynamics applications.

Swept
uses algorithms which resolve swept configurations better.

Advanced
uses algorithms which can decompose the volume into a combination of mapped, swept and unstructured blocks. It automatically imprints free faces to aid with sweeping, etc. This will give

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a result similar to the Multi-zone Method found in ANSYS Meshing. Some work decomposing the geometry or surface blocking beforehand can improve the results.

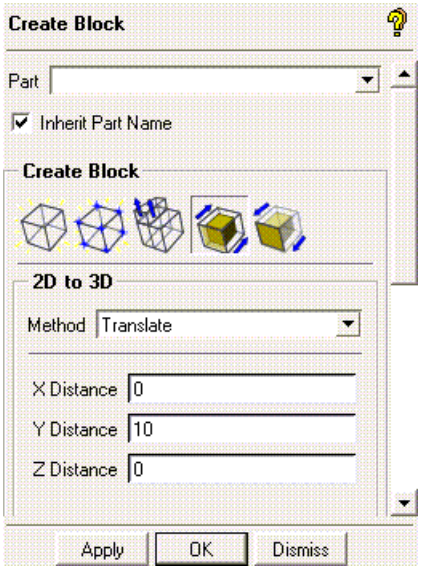
Note

Behind the scenes several possible algorithms are run to create the 3D blocking. One such algorithm is similar to converting each surface block into one element, with Mapped blocks converted into quads; then in an operation similar to the Hex dominant mesher, the volume is filled with hexa elements. If this is successful, then these are converted back into 3D blocking to replace the original 2D surface blocking. Other algorithms use the mapped side mesh to guide the sweeping of unstructured faces into swept blocks (similar to the Cooper tool). Any remaining volumes are simply declared as unstructured blocks and can be filled with one of the bottom up options, including Tetra, Hexa Core or Hexa Dominant mesh.

Translate

allows you to specify the X, Y and Z distance to extrude the 2D Block. When **Inherit Part Name** is enabled, the 3D block inherits the blocking material (part) of the original 2D block.

Unstructured 2D blocks will be converted to Swept blocks on translation.



Rotate

allows you to extrude a 2D Block by rotation. When **Inherit Part Name** is enabled, the 3D block inherits the blocking material (part) of the original 2D block.

Unstructured 2D blocks will be converted to Swept blocks on rotation. Degenerate (axis-collapsed) swept blocks can also be handled. The degenerate 7–node hexa elements on the axis can be converted to regular elements using the **Write 7–node-hexas as pyramids** option in the **Hexa/Mixed Meshing Options** DEZ (see Hexa Meshing (p. 90)).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

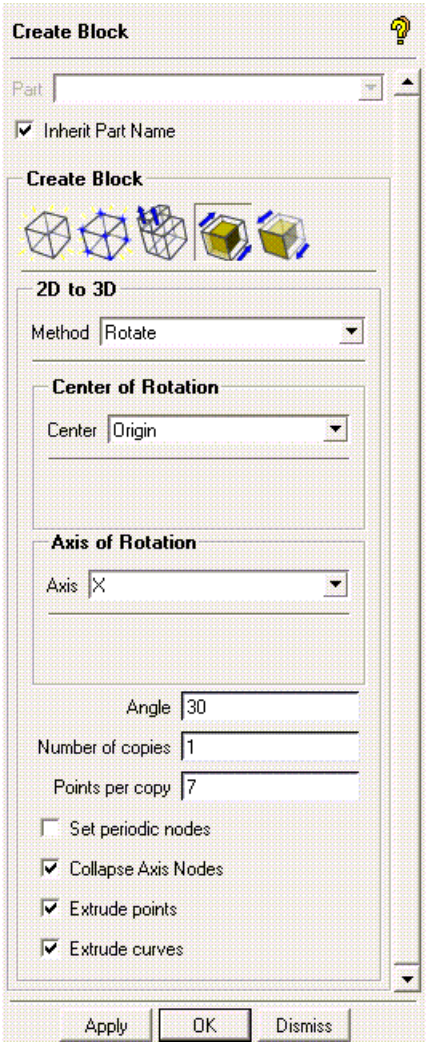
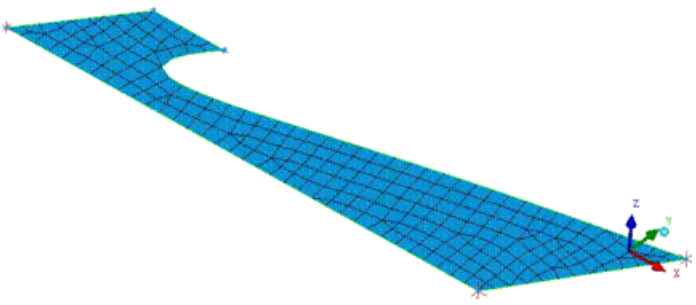
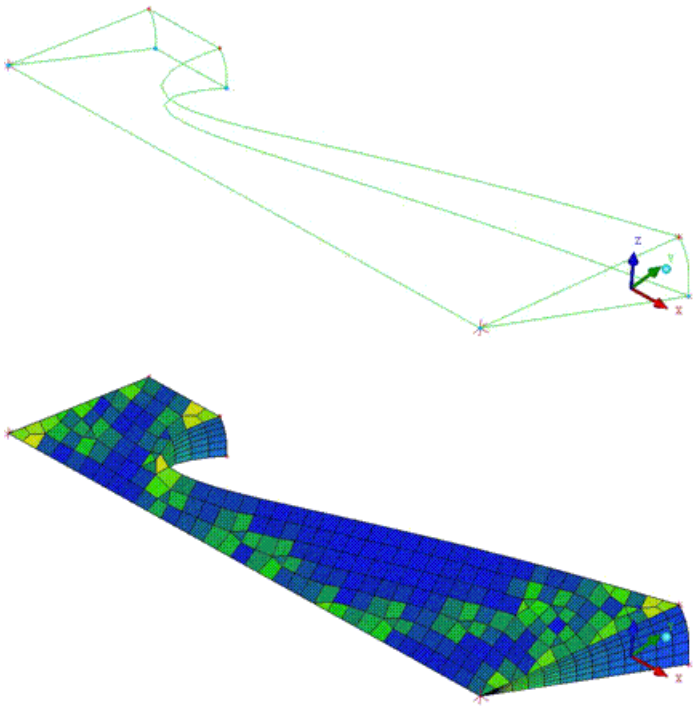


Figure 335: 2D to 3D Rotate for a 2D Unstructured Block (p. 369) shows an example of a 2D unstructured block extruded using the 2D to 3D rotate option using an **Angle** of 30 degrees, **Number of copies** set to 1, and **Points per copy** set to 7 as shown in the DEZ above.

Figure 335: 2D to 3D Rotate for a 2D Unstructured Block



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Center of Rotation
specifies the center of rotation. The **Origin** option allows you to rotate the block in the specified axis direction about the origin (0,0,0). The **User** option allows you to rotate the block about a specified point.

Axis of Rotation
specifies the axis or vector about which to rotate the block.

Angle
specifies the angle of rotation for each block copy.

Number of copies
specifies the number of copies of blocks.

Points per copy
specifies the number of nodes on the extruded edges.

Set Periodic Nodes
defines periodicity at both the geometry and blocking level, and will override any existing periodicity information set under Global Mesh Setup > Set Up Periodicity.

Collapse Axis Nodes
allows you to collapse all the nodes lying on the axis of rotation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extrude points, Extrude curves
when enabled, the original 2D geometry gets converted to 3D as well. Points are extruded to 3D curves and curves are extruded to 3D surfaces.

> **Note**
>
> The node to point association must be defined before extrusion in order for points to be extruded to produce 3D curves. Similarly, the edge to curve association must be defined for the curves to be extruded to 3D surfaces.

3D to 2D

The **3D to 2D** option converts 3D Blocking into 2D Blocking, as shown in Figure 336: 3D Blocking (p. 371) and Figure 337: 2D Blocking (p. 371).

Figure 336: 3D Blocking

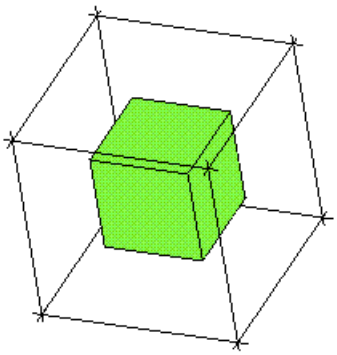
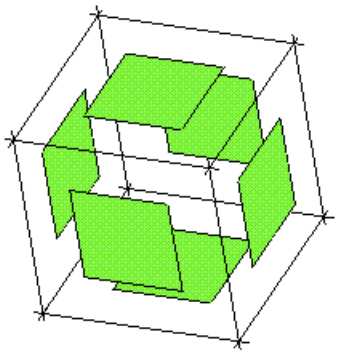


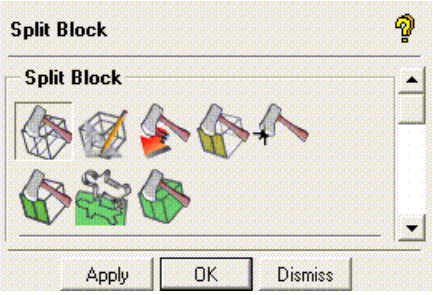
Figure 337: 2D Blocking



Split Block

The **Split Block** option allows you to make multiple blocks from a single block.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



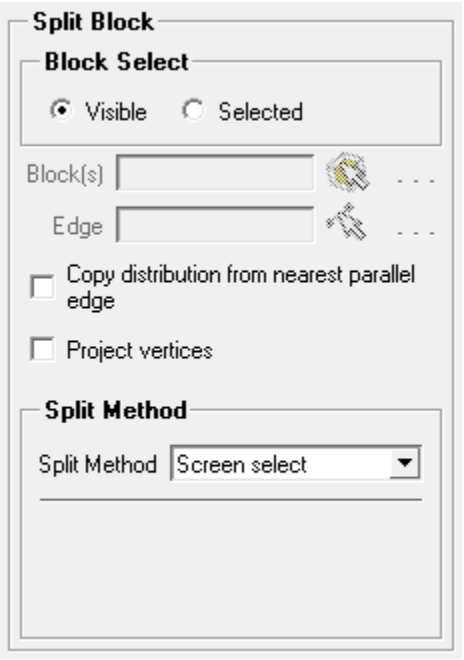
The following options are available for splitting blocks:

Split Block
Ogrid Block
Extend Split
Split Face
Split Vertices
Split Free Face
Imprint Free Face
Split Free Block

Split Block

The **Split Block** option splits the blocking at the selected edge.

Figure 338: Split Block Options



Visible
splits only blocks that are visible, where visibility is managed by blanking blocks in the graphical window or by using the index control.

Selected
splits only the selected blocks.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy distribution from nearest parallel edge
copies the edge distribution (initial heights, ratios and bunching law) from the nearest parallel index to the new edges created by the split operation. This is useful when adding splits to a blocking after already setting up advanced distributions on existing edges. With this option disabled, the default edge distribution (**Settings > Meshing Options > Hexa/Mixed**) is used. Parallel node counts are always used when splitting a block.

Project vertices
enables the automatic projection of the new vertices to the underlying geometry.

Split Method

Screen select
allows you to select the position manually where the block is to be split.

Prescribed point
allows you to split the edges through the selected point.

Relative
allows you to split the edge with the given parameter.

Absolute
allows you to split the edge in proportion to the maximum grid length in the edge direction. Edge direction corresponds to minimum vertices number to maximum vertex number.

Curve parameter
allows you to split the edge through a selected point on a curve. Select the edge to be split first, and then the curve at the point through which the split will be placed.

Note

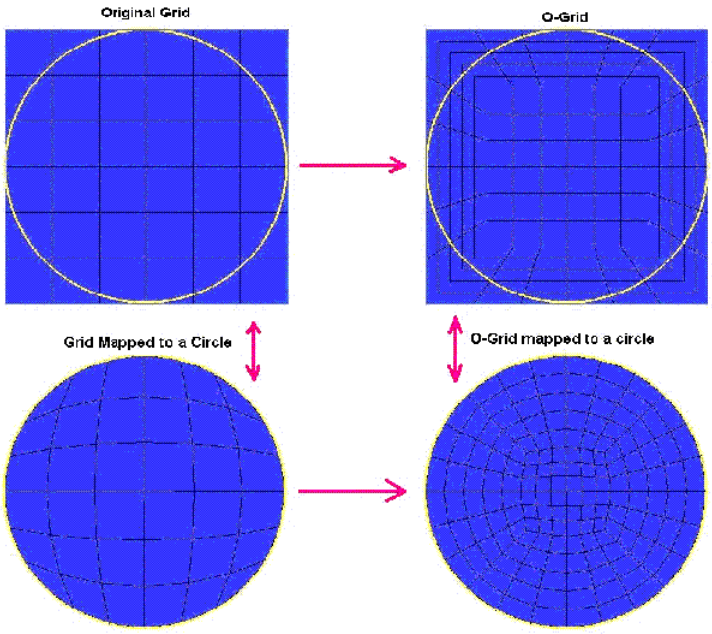
The **Split Block** option will only propagate splits through mapped faces, and will terminate at free faces. For 2D blocking, all mapped blocks attached edge to edge will be split, but the split will terminate at any free block. For 3D blocking, a split will propagate through mapped blocks and swept blocks, if the split is along the sweeping direction. A split in 3D blocking will terminate at free 3D blocks and swept blocks, if the split is in any direction other than the sweeping direction.

Ogrid Block

The **Ogrid Block** option allows you to modify a single block or blocks to a 5 sub-block topology (7 sub-blocks in 3D) as shown below. It arranges grid lines into an “O” shape to reduce skew where a block corner lies on a continuous curve or surface. There are several variations of the basic O-grid generation technique.

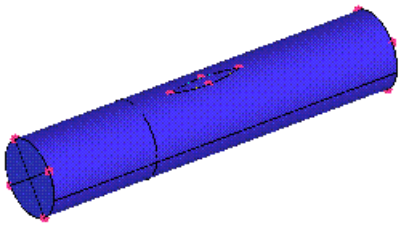
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 339: O-grid Demonstration



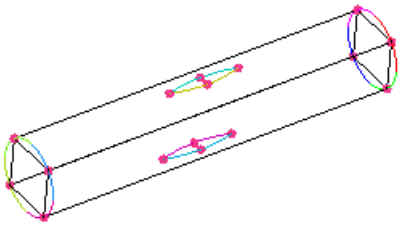
The O-grid can be created with or without face selection as shown in Figure 340: O-grid Creation With and Without Face Selection (p. 374).

Figure 340: O-grid Creation With and Without Face Selection



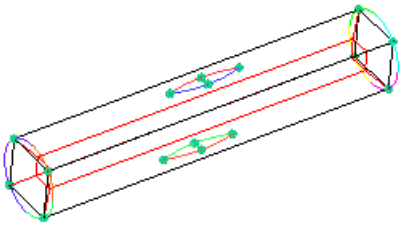
a) Circular Cylinder

b) Block Inside Cylinder

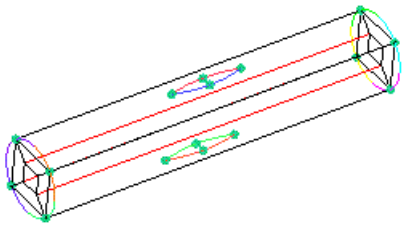


c) O-grid created without face selection

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

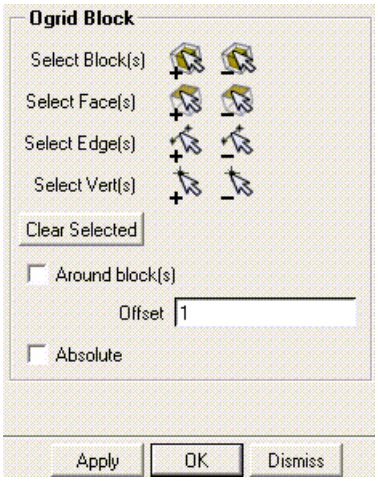


d) O-grid created with face selection



O-grids can be created or removed by selecting or deselecting Blocks, Faces, Edges, and Vertices.

Figure 341: Ogrid Creation Options



The following options are also available:

Clear Selected
clears the previously selected entities.

Around block(s)
enables the creation of the O-grid around the block(s).

Offset
specifies the height of the O-grid layer.

Absolute
when enabled, the value assigned for **Offset** is the actual length of the radial edge of an O-grid. Assigning a **Offset** of 7 would make all the radial edges of the O-grid 7 units in length.

When **Absolute** is disabled, **Offset** behaves like a relative distance: A value of 1 causes the Hexa Mesher to place the O-grid at a location where the resulting blocks will be distorted the least. A

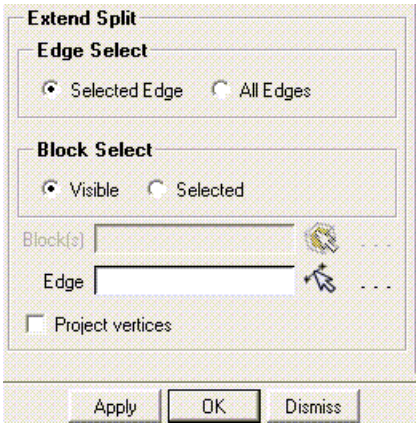
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

higher value makes the inner blocks of the O-grid smaller, with the surrounding blocks larger, and a smaller value makes the inner blocks larger and the surrounding blocks smaller.

Extend Split

The **Extend Split** option allows you to extend block splits to a selected edge or to all edges. You can select specific blocks or all visible blocks to extend the split.

Figure 342: Extend Split Options

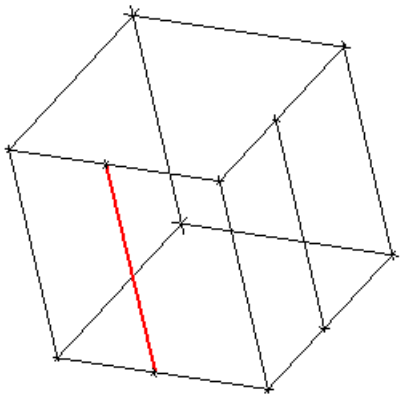


Edge Select

Selected Edge

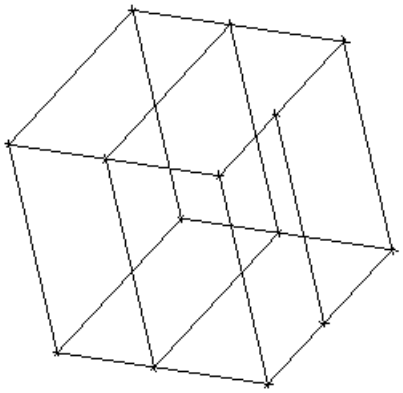
when selected, allows you to select an edge to extend the split (see Figure 343: Edge Selected to Extend Split (p. 376) and Figure 344: Block Split Extended to One Edge (p. 377)).

Figure 343: Edge Selected to Extend Split



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 344: Block Split Extended to One Edge



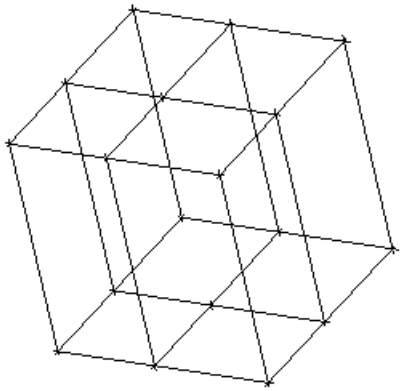
Block Select
contains options for selecting blocks when the **Selected Edge** option is selected.

Visible
when selected, the split will be extended through all visible blocks.

Selected
when selected, allows you to select specific blocks through which the split will be extended.

All Edges
when selected, the block split will extend to all the edges (see Figure 345: Block Split Extended to All Edges (p. 377)).

Figure 345: Block Split Extended to All Edges



Project vertices
enables the automatic projection of the new vertices to the underlying geometry.

Split Face

The **Split Face** option allows you to split a face at a desired location. Select the edge where the split will take place and hold down the left mouse button to slide the cursor on that edge to the desired split location. Release the left mouse button and press the middle mouse button to accept the location

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and to complete the face split operation. See Figure 346: Face Selected for Face Split (p. 378) and Figure 347: Face Split (p. 378).

Figure 346: Face Selected for Face Split

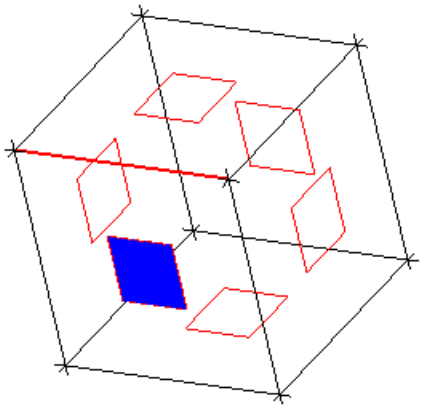
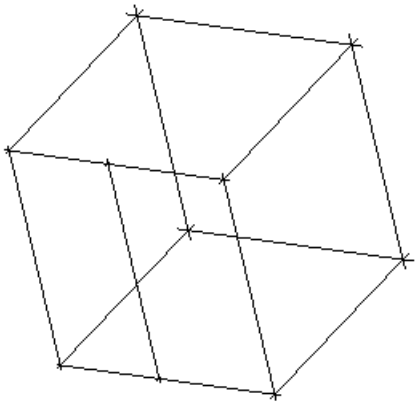


Figure 347: Face Split



You can select multiple faces to split by utilizing the corner vertices tool from the selection toolbar.

Split Vertices

The **Split Vertices** option allows you to split a degenerate vertex. This is equivalent to uncollapsing an edge.

The block and vertex shown in Figure 348: Vertex Selected to Split (p. 379) is the result of collapsing a single block. Select the appropriate vertex to split, and click **Apply** to split the vertex as shown in Figure 349: Split Vertex (p. 379)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 348: Vertex Selected to Split

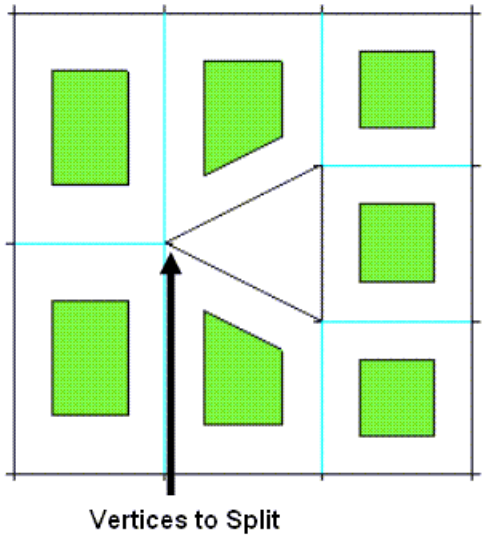
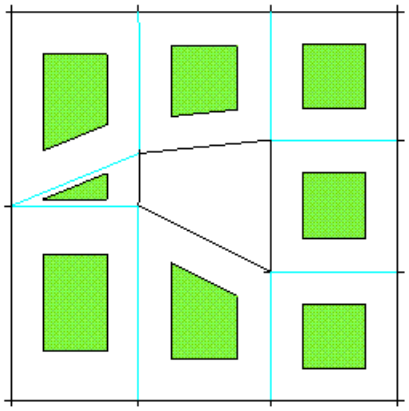


Figure 349: Split Vertex



Split Free Face

The **Split Free Face** option allows you to split the free face of a block, which splits the block into two free blocks.

Imprint Free Face

The **Imprint Free Face** option allows you to imprint edges from one face onto an unstructured 2D surface block (face).

Note

This task is automatically accomplished to sweep blocks when the **Create Block > 2D to 3D > Fill > Advanced** option is used; but it may sometimes be advantageous to do it manually before using the 2D to 3D fill or to imprint edges for other reasons, such as node for node matching between free faces for connection, or periodicity.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

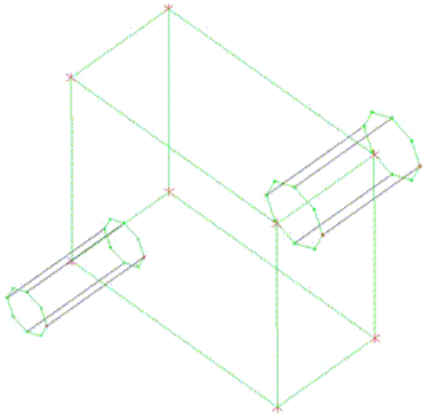
Free Block
specifies the selected target block. Usually this is an unstructured 2D face across from the edges you want to imprint.

Loop Vertices
specifies the vertices that form the loop of edges you want to imprint on the free block.

An example demonstrating the use of the **Imprint Free Face** option is shown in Figure 350: Use of the **Imprint Free Face** Option (p. 380).

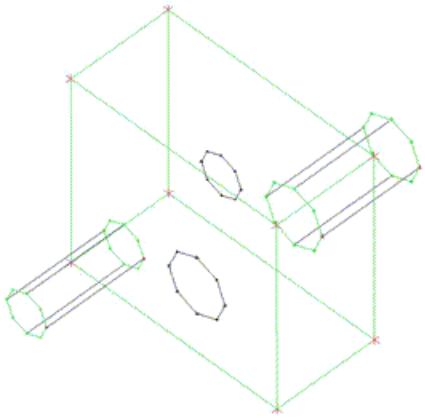
Figure 350: Use of theImprint Free Face Option

(A) The 2D surface blocking for the geometry



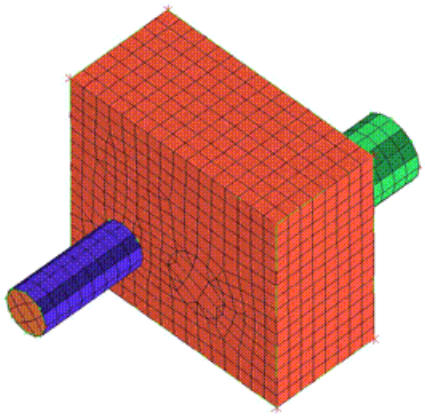
The central block cannot be swept because it has different features on either side.

(B) The 2D surface blocking with the faces imprinted on either side of the central block



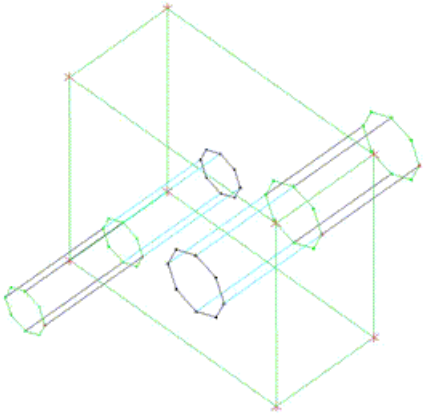
(C) The imprinted face meshed

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



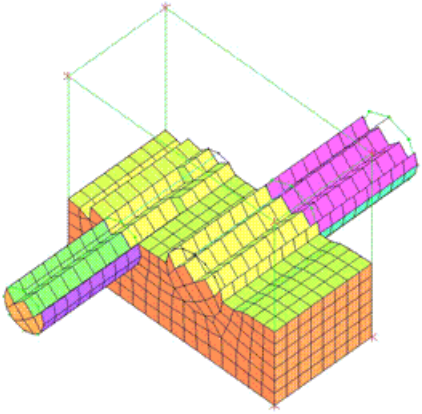
The imprinted face is a free mesh, even though the source mesh (circle) was mapped.

(D) The filled model



The 2D blocking is converted to 3D blocking using the **Fill** method and **Fill Type** set to **Swept** for **Blocking > Create Block > 2D to 3D**.

(E) The filled model meshed



Note that the imprinting improved the sweeping.

Split Free Block

The **Split Free Block** option allows you to decompose a free block further into structured and swept blocks. This operation requires that you first create sheet block(s) which will then be used to split the free block. A free block can be split into multiple volumes in a single operation. You can then convert the split free blocks to structured or swept blocks as required.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sheet blocks used to split the free block are internal faces which pass through the volume. Only structured sheets are currently supported. A structured sheet can be created from 4 vertices or points (see the Sheet (p. 361) option in the 3D Blocks (p. 357) section). The split sheets can be either contiguous or separated by other faces. However, the selected split sheets combined with other faces of the free block should split the volume into manifold regions. Only those sheets which form the closed volume need to be specified.

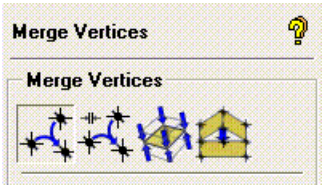
Note

- Currently, the sheet faces must be structured. However, other faces of the free block to be split can be of any type (structured or free).
- Free blocks with multiple shells can be split only if the split sheets connect inner and outer shell faces. This will result in single shell manifold volumes after the split.
- Split sheets must be connected to the boundary faces, a rectangular region fully embedded inside the free block is not supported.

Merge Vertices

The **Merge Vertices** menu offers the user options for combining two or more points.

Figure 351: Merge Vertices Options



The following options are available for merging vertices:
Merge Vertices
Merge Vertices by Tolerance
Collapse Block
Merge Vertex to Edge

Merge Vertices

The **Merge Vertices** option allows you to merge two or more vertices.

Two vertices can be merged with the following options:

Propagate merge
propagates the merged vertices throughout the affected block(s).

Merge to average
merges at the average distance of the two vertices.

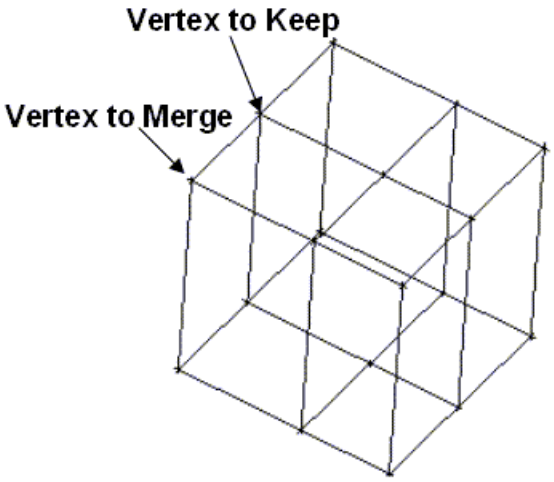
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rebuild orphan

deletes the VORFN part and rebuilds it. This may be necessary in some cases to clean up the VORFN blocks.

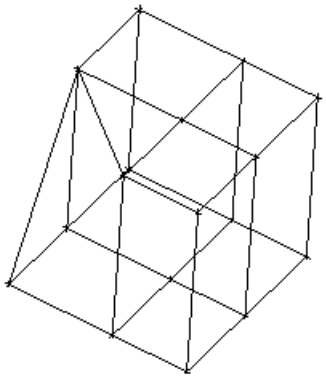
These options can be used in different combinations as illustrated for the example of the two vertices to be merged in Figure 352: Selection of Vertices to be Merged (p. 383). When selecting the vertices, the first vertex is retained and the second vertex is merged with the first.

Figure 352: Selection of Vertices to be Merged



- **Propagate merge** and **Merge to average** are disabled. Only the selected vertices will be merged (see Figure 353: Merge Vertices Without Options (p. 383)).

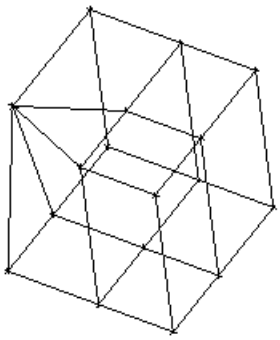
  Figure 353: Merge Vertices Without Options



- **Propagate merge** is disabled while **Merge to average** is enabled. Only the selected vertices will be merged at the average distance of the two (see Figure 354: Merge to Average Option Only (p. 384)).

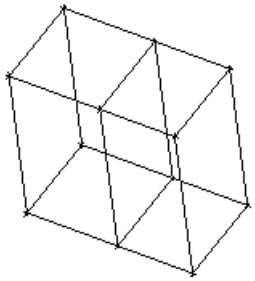
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 354: Merge to Average Option Only



- **Propagate merge** is enabled and **Merge to average** is disabled. All the connected vertices will be merged according to the indices value (see Figure 355: Propagate Merge Option Only (p. 384)). For example, the **Vertex to Keep** has indices (x1, y, z) and **Vertex to Merge** has the indices (x2, y, z) so all the vertices having indices (x1,*,*) will merge with the same family of vertex having the indices (x2,*,*) for example a vertex with indices (x1,y1,z1) will only merge with vertex with indices (x2,y1,z1).

Figure 355: Propagate Merge Option Only



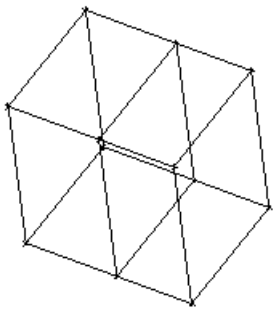
Note

After selecting the vertices, a **Confirm delete station** window will ask you to confirm the direction and the index range to delete the vertices to be merged.

- **Propagate merge** and **Merge to average** are enabled. All the connected vertices will be merged similar to the previous case, but at an average distance between the selected vertices (see Figure 356: Both Propagate Merge and Merge to Average Options Selected (p. 385)).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 356: Both Propagate Merge and Merge to Average Options Selected



Merge Vertices by Tolerance

The **Merge Vertices by Tolerance** option allows you to merge only those nodes that come under the specified tolerance limit. Select the vertices to be merged, and specify the tolerance value.

Collapse Block

The **Collapse Block** option allows you to collapse a block or an edge by shrinking one or more edges to zero length. This modifies the topology of the remaining non-degenerate blocks. Select the edge to define the direction that is to be collapsed to zero size. It is possible to add blocks interactively, which will all be collapsed in the specified direction.

In Figure 357: Selection of Blocks and Edge for Collapse (p. 385), the edges and blocks selected to be collapsed are shown in red. Figure 358: Collapsed Blocks (p. 385) shows the collapsed blocks.

Figure 357: Selection of Blocks and Edge for Collapse

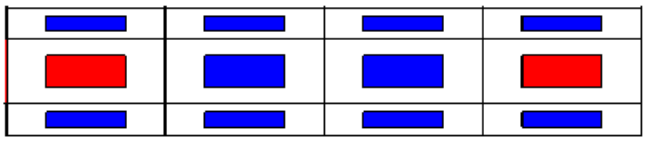
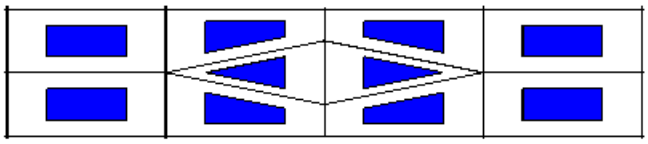


Figure 358: Collapsed Blocks



Merge Vertex to Edge

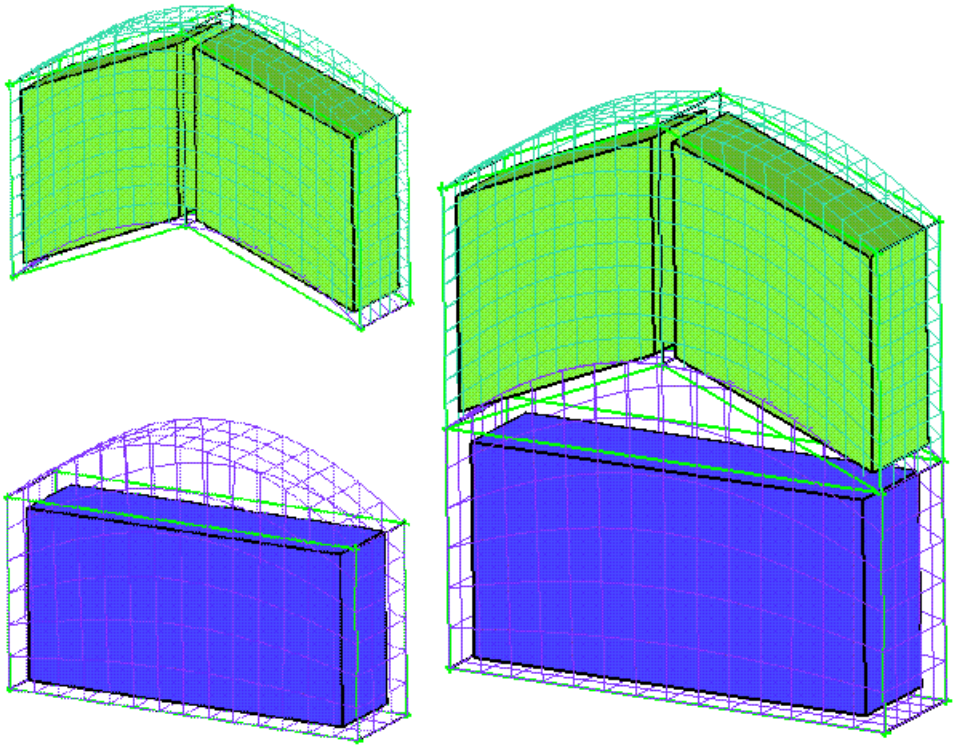
The **Merge Vertex to Edge** option allows you to merge a selected vertex and edge. The selected edge will be split and merged with the selected vertex.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is most often utilized when the region between the edge and vertex contains no blocks, and the two disconnected topologies which do not have matching vertices can be merged. Examples of this would include connecting bottom up blocking or merging blocking topologies.

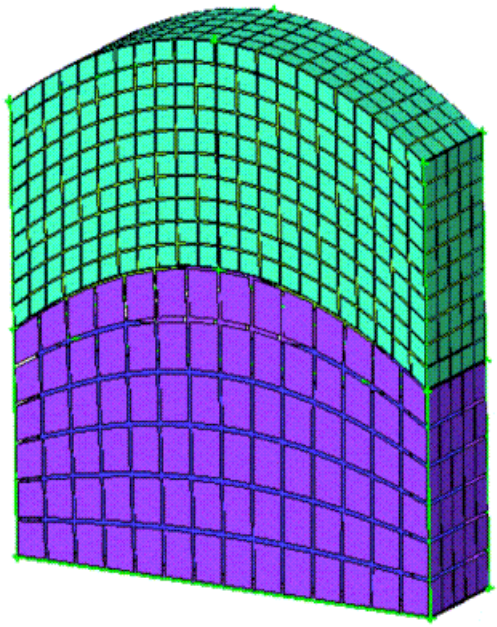
Figure 359: Connected Blocking Topologies (p. 386) shows two sub topologies which are connected at the corners but not the middle. The resulting surface mesh is shown in Figure 360: Resulting Surface Mesh — Unmatched (p. 387). The top of the lower part is curved to fit the upper part as the edges are projected to the same curves. However, as the edges are not shared, the mesh across these edges is neither connected nor matched.

Figure 359: Connected Blocking Topologies



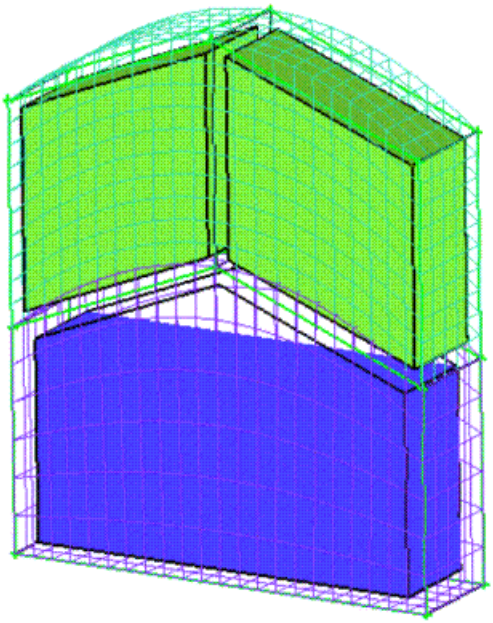
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 360: Resulting Surface Mesh — Unmatched



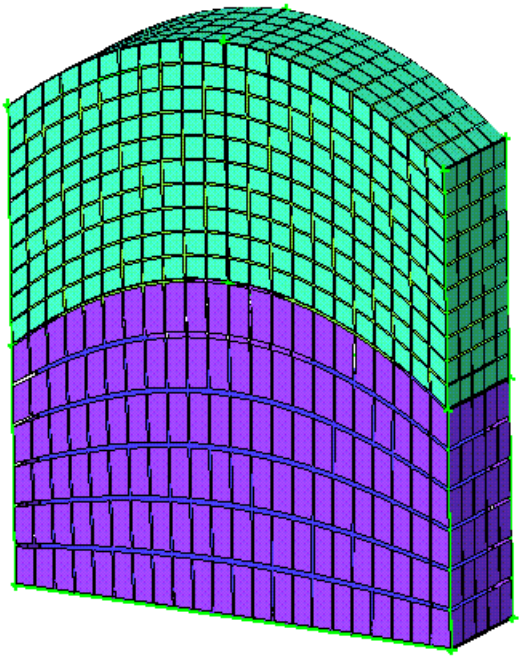
The lower block can be split to provide the matching vertices and then the **Merge Vertices** function could be used, or the **Merge Vertex to Edge** function can be used to merge the "hanging" vertices directly to the edge. This will split the edges for the connection, but the block will still appear as a single block (see Figure 361: Vertex Merged to Edge (p. 387)). After merging the vertex with the edges, the surface mesh will be properly connected (see Figure 362: Resulting Surface Mesh — Matched (p. 388)).

Figure 361: Vertex Merged to Edge



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

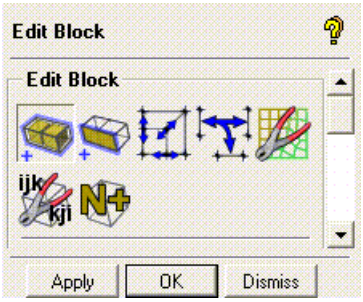
Figure 362: Resulting Surface Mesh — Matched



Edit Block

Use the **Edit Block** menu to modify existing blocks.

Figure 363: Edit Block Options



The following options are available for editing blocks:

Merge Blocks
Merge Faces
Modify Ogrid
Periodic Vertices
Convert Block Type
Change Block IJK
Renumber Blocks

Merge Blocks

The **Merge Blocks** option allows you to merge several blocks into a single large block.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selected
merges all the selected blocks.

> **Note**
>
> When merging *free* blocks, you may merge only two, 2D free blocks at one time. You may not merge 3D free blocks or mix mapped and free blocks.

Automatic
automatically merges all the blocks in the current Block part to the least number of blocks possible.

IJK Direction
merges the blocks in the direction determined by the reference edge selected.

Merge Faces

The **Merge Faces** option allows you to merge faces and their corresponding blocks. Select the faces to be merged by selecting the face corners.

Modify Ogrid

The **Modify Ogrid** option allows you to modify the scale factor of an O-grid.

Method
specifies the method for modifying the O-grid.

Rescale Ogrid
allows you to rescale the O-grid.

Block Select
allows you to select all visible blocks or specific blocks.

Edge
is the radial O-grid edge selected.

Absolute distance, Offset
control the rescaling of the O-grid. If **Absolute distance** is disabled, the **Offset** value works as a factor multiplied by the current O-grid size, where values less than 1 result in a smaller O-grid, while values greater than 1 yield a larger O-grid. If **Absolute distance** is enabled, the **Offset** value is the new length of the radial edge of the O-grid.

Reset Ogrid Orthogonality
allows you to reset the orthogonality of the selected edge. When an O-grid is created, a radial set of edges connect the internal/outer O-grid vertices to the external/inner O-grid vertices. The vertices are orthogonal to each other along the radial edge. This orthogonality may be modified when vertices are moved.

Periodic Vertices

The **Periodic Vertices** option allows you to make selected pairs of vertices into periodic nodes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create
allows you to select pairs of nodes that should be periodic transforms of one another. A face is a periodic transform of another if its four corners are periodic transforms of the other face's nodes. When a periodic face is split, all the new vertices and faces will be periodic.

Note

An axis node should be selected twice in order to make it periodic with itself.

Remove
allows you to select a pair of vertices to remove the periodicity.

Note

To generate a periodic mesh, remember to make the initial block periodic. Further splits will maintain this periodicity. It is much easier to make the first four pairs of vertices periodic then to match up many vertices later to be made periodic.

Auto Create
creates periodic links between block vertices on the periodic boundaries automatically.

Note

This works on a 3D Blocking with model periodicity defined. It finds the periodic twins through Geometry transformation and blocking edge connectivity. Periodic vertices that are associated to points are found directly by geometry transformation. It expects that periodic points are within a geometric tolerance of 1.0e-05. Periodic vertices that are projected to curves are found through periodic curve pairs or blocking edge connectivity along the periodic direction. Surface associated vertices are first found by transformation or best closest match.

Convert Block Type

The **Convert Block Type** option allows you to convert blocks into the following blocking types. See User's Manual > Hexa > Hexa Block Types for a description of the different block types.

Mapped
converts a block to a mapped block. The block converted must have 4 corners in 2D (or 8 corners and 6 mapped faces in 3D). The Mapping algorithm forces the number of nodes on opposite sides to be equal. If they are not equal, the larger number will be used. This number of nodes will be propagated through all parallel edges.

Free
converts a block to a free (unstructured) block. In 2D, this will result in a paved surface which can be all triangles, quad dominant, quad with one tri, or all quad. In 3D, the free block mesh type can be Tetra (Delaunay, TGrid or Advancing Front), Hexa Core or Hexa Dominant mesh. In 2D, this is most often used

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

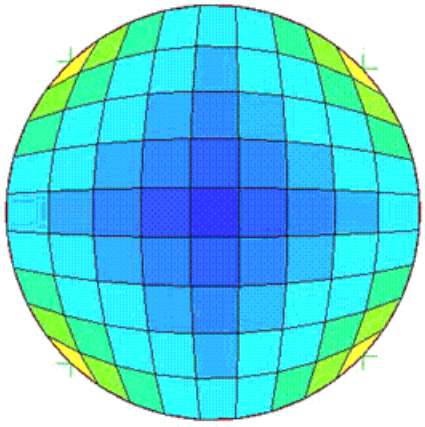
to solve issues with poor internal angles after mapping to curves or to prevent node counts from propagating across a face.

Note

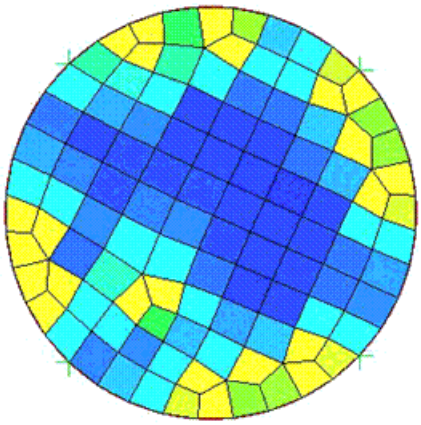
An error will be reported if the selected block is in the VORFN part.

Figure 364: Conversion of a Mapped Block to a Free Block

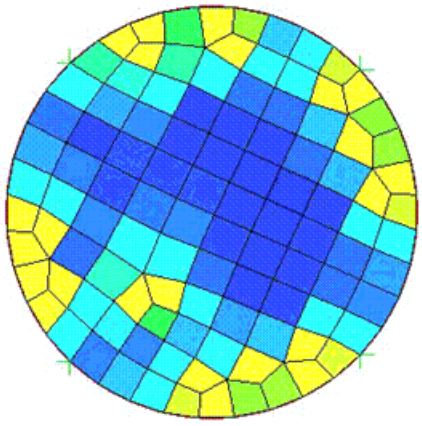
(A) Mapped Block



(B) Free Block with All Quad Mesh

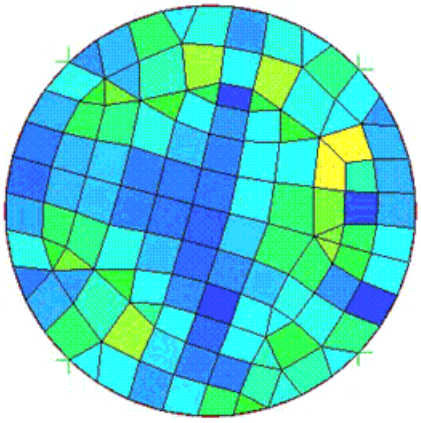


(C) Free Block with Quad with one Tri Mesh

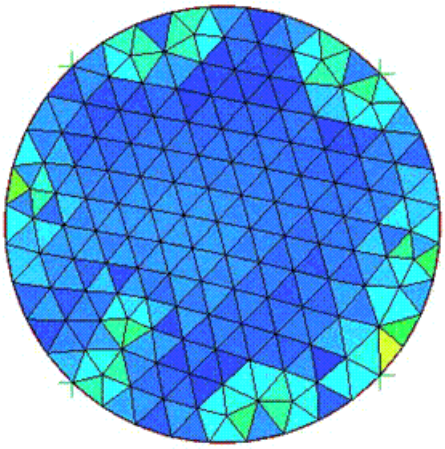


ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(D) Free Block with Quad Dominant Mesh



(E) Free Block with All Tri Mesh



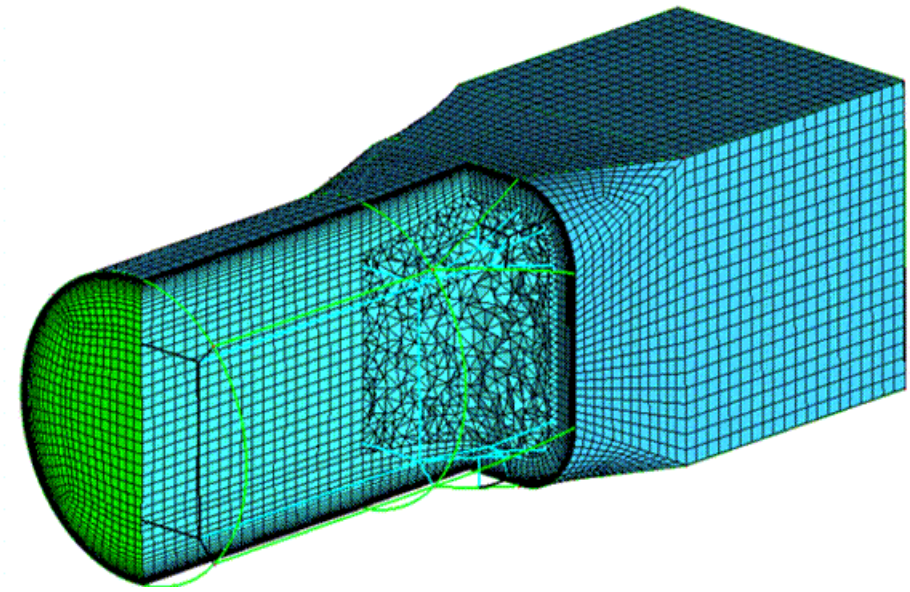
Swept

converts a mapped block to a swept block. The face selected is converted to a free face and swept across the block and any parallel blocks. While mapped blocks must have the same nodes on opposite sides of the block in the I, J and K directions, a swept block is free to have varying nodes in two directions (across the selected face) and is mapped in the third (perpendicular to the selected face).

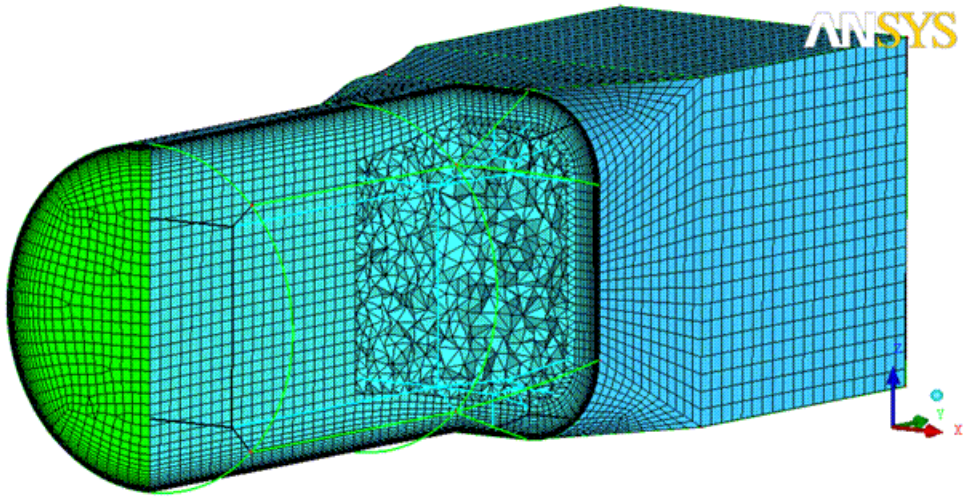
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 365: Conversion of a Mapped Block to a Swept Block

(A) Mapped Block



(B) Swept Block



Y-Block

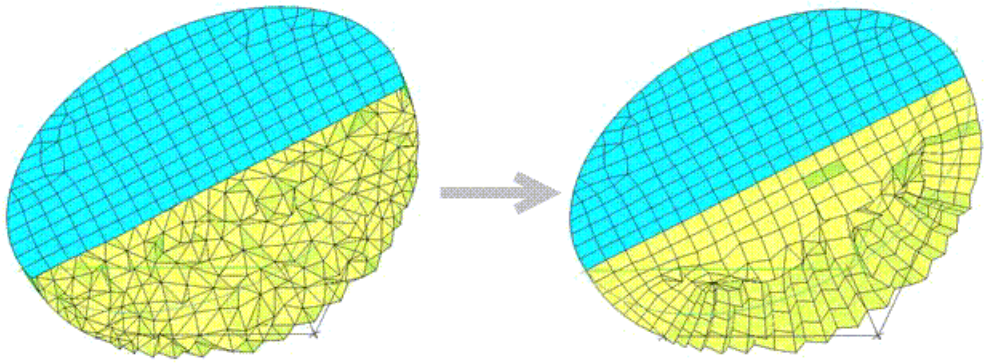
converts a degenerate (wedge) block to a Y-block (quarter O-grid).

3D Free Block Mesh Type

converts a free block mesh to an unstructured mesh type of Tetra or Hexa-Core mesh. Figure 366: Changing the 3D Free Block Mesh Type (p. 394) shows the conversion of the unstructured Delaunay tetra mesh (with a Free Quad Dominant face) to hexa-dominant mesh using this option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

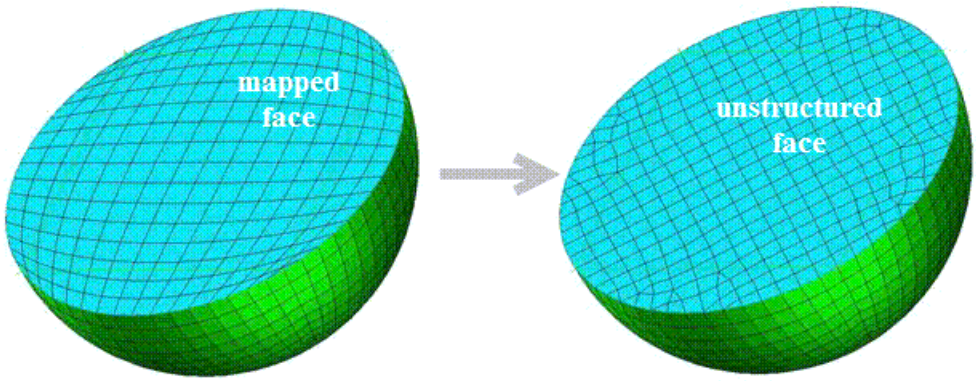
Figure 366: Changing the 3D Free Block Mesh Type



Convert free block face to free

allows you to convert the mapped external face adjacent to a free block to a free face. In Figure 367: Converting Free Block Face to Free (p. 394), the circular face adjacent to the hexa mesh is initially mapped. The unseen volume is free (unstructured tetra in this case). This results in poor elements at the corners (the maximum angle approaches 180 degrees as the mesh is refined). Using this option to convert the structured face to an unstructured face (quad dominant in the example) results in improved quality.

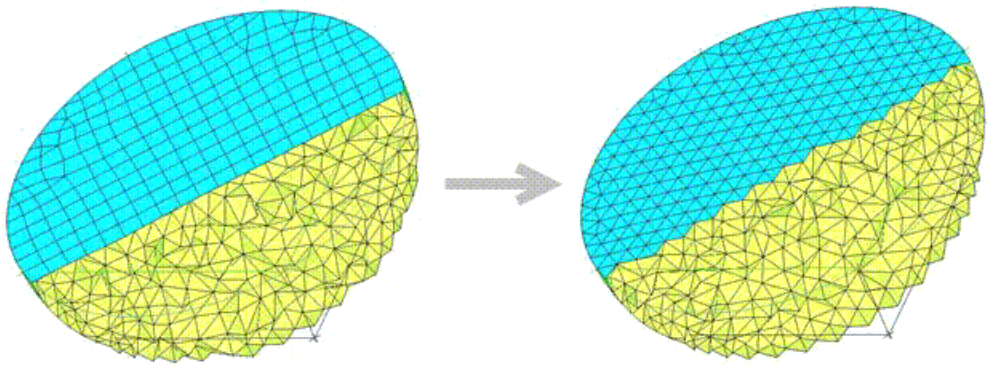
Figure 367: Converting Free Block Face to Free



Free Face Mesh Type

converts the free face mesh type to the type selected. Figure 368: Changing the Free Face Mesh Type (p. 394) shows the conversion of the free face mesh type from the default quad-dominant to the all tri type which is more appropriate for the tetra mesh in the adjacent free block.

Figure 368: Changing the Free Face Mesh Type



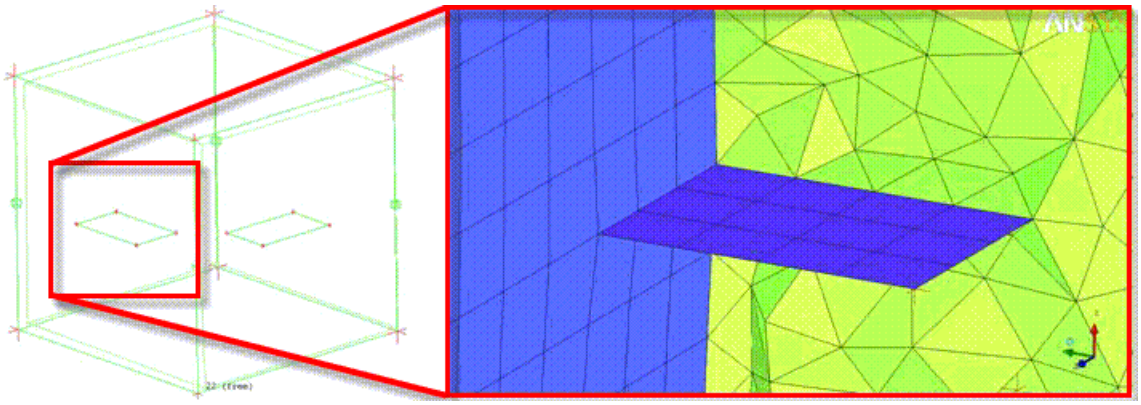
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverse Swept Direction
reverses the sweep direction for a swept block. This can be useful in some situations where you want to better control the seeding on the free face of the sweep, such as when using a reference mesh (see the **Reference Mesh** option for the **Associate Face to Surface** option).

Merge Sheet with free Block
allows you to merge a 2D (sheet) block with a 3D free block. Using this option will ensure node to node connections between the two regions. For CFD users, this option can be used to connect a wall-mounted baffle with the rest of the mesh. For FEA users, this option can be used to combine 2D and 3D portions of the geometry.

Figure 369: Merging 2D (Sheet) Block With a Free Block (p. 395) shows an example where a wall-mounted baffle has been connected to rest of the mesh using this option.

Figure 369: Merging 2D (Sheet) Block With a Free Block



Change Block IJK

The **Change Block IJK** option allows you to change the IJK indices of a block.

IJK->KIJ
changes indices from IJK to KIJ.

Set Origin
sets the origin at a selected vertex.

Align Blocks
aligns all blocks with reference to a selected block.

Set IJK
sets the current IJK indices to new IJK indices.

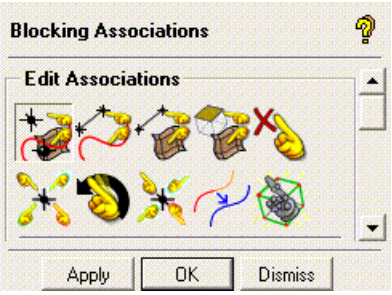
Renumber Blocks

The **Renumber Blocks** option allows you to renumber blocks.

Associate

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 370: Blocking Associations Options



The following options are available for blocking associations.

Associate Vertex
Associate Edge to Curve
Associate Edge to Surface
Associate Face to Surface
Disassociate from Geometry
Update Associations
Reset Associations
Snap Project Vertices
Group/Ungroup Curves
Auto Associate

Associate Vertex

The **Associate Vertex** option allows you to associate vertices and project the vertex onto itself, points, curves, and surfaces. Select the vertex and the entity to project it onto.

Associate Edge to Curve

The **Associate Edge to Curve** option allows you to associate the edges of blocks to curves. The vertices at the end of the edges are also associated to the same curve unless they were previously associated to another curve or point. Edge segments can be individually associated after using edge splits. Multiple edges can be associated with multiple curves, but all the curves will be grouped into a single composite curve.

Note

Associating edges to curves also results in the creation of line elements along those curves. For 2D planar blocking, it is essential that all the perimeter edges be associated with perimeter curves because many solvers use the perimeter line elements as boundaries.

Project vertices

If enabled, the vertices will automatically be projected to the corresponding curves.

Project to surface intersection

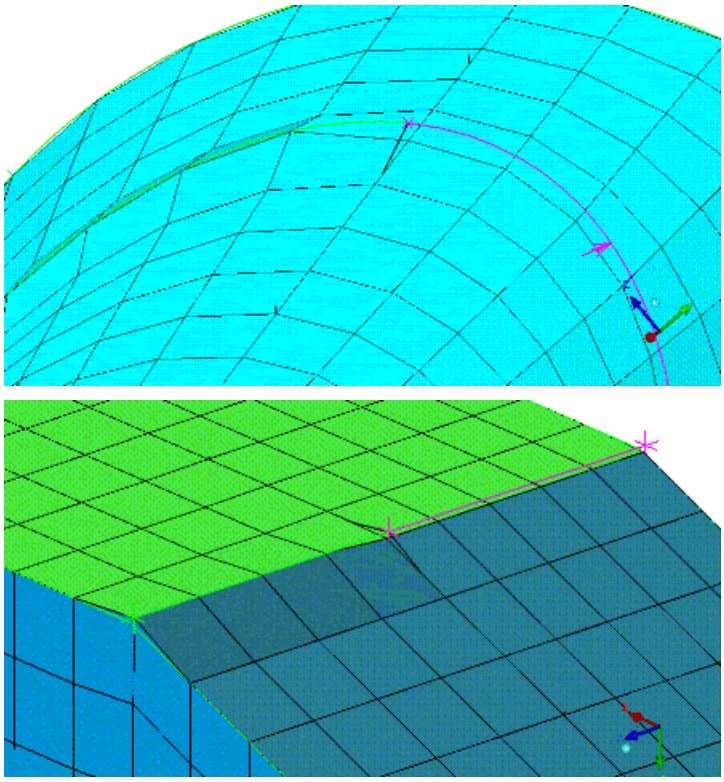
If enabled, the surface-surface intersection will be captured correctly. This is for poor geometry situations where the intersection curve may not match with the intersection of the surfaces. If you associate an edge with a curve using this option, the edge will be colored purple. The edge is associated with the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

curve, but when the mesh is generated, the nodes will first project to the curve, and then they will project to the surface (the true surface/surface intersection).

In the examples in Figure 371: Examples of Project to Surface Intersection (p. 397), the intersection curve is slightly above the surface. The green edge on the left is associated with the curve without enabling this option and therefore its nodes are off the true surface intersection. The purple edge on the right is associated to the curve with this option. The edge is therefore associated with the curve, but the nodes ultimately project to the surface intersection. This is better than simply projecting the edge to the curve because the nodes are guided by the curvature of the curve.

Figure 371: Examples of Project to Surface Intersection



Project ends to curve intersection

If enabled, the vertices will be enforced at the ends of the curve.

Associate Edge to Surface

The **Associate Edge to Surface** option allows you to associate the edges of blocks to surfaces. The color of the selected edges will turn white/black, indicating that the nodes on that edge will be projected to the nearest active surface. By default, edges and faces between two blocking materials (boundary faces) will automatically associate with the nearest active surface. Related vertices and edge splits will also be associated with the nearest active surface. When moving vertices or edge splits, there will be no movement until the cursor passes over an active surface. Ensure that the vertices or edge splits are moving on the correct surface.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Associate Face to Surface

The **Associate Face to Surface** option allows you to adjust how faces relate to the geometry. By default, edges and faces between two blocking materials (boundary faces) will automatically associate with the nearest active surface.

Click the selection icons to select the face(s) and the surface part.

The following options are available for associating faces to surfaces:

Closest
finds the closest surface for projection. For boundary faces, this returns face association to the default. It can also be used to force internal faces to follow a surface.

Interpolate
interpolates the shape of the meshed face from the shape of the bounding edges, rather than projecting the nodes to the surface. The surface mesh will still inherit the family name of the closest surface part. Use this feature if a face crosses a missing or poor quality surface.

Part
projects the face to the surface that is in the specified part. This is useful for closely spaced curved surfaces, such as turbine blades, to ensure that each face projects to the correct surface, and not merely the closest surface.

Note

If you associate the face to a part which contains no surfaces, it will instead interpolate the mesh and place the elements in the specified part. This can be used to control the part name of the interpolated mesh.

Shared Wall
allows you to set a general projection rule regarding faces between two specific volume parts.

Create
sets the rule that the faces between the specified volume parts should always be projected to surfaces of the specified surface part.

Remove Shared Wall
removes the above rule for a given pair of volume parts.

None
removes the default behavior so that edges/faces between specified volumes do not automatically associate with surfaces. This is useful when there is no boundary between volumes or if you are using volume parts to break up a model for display or selection purposes, but the regions don't actually represent physical differences which would require a boundary.

Link Shapes
allows the internal face to have the same shape as the linked boundary face. The faces can be unlinked by disassociating them. Select the boundary face and the associated internal face.

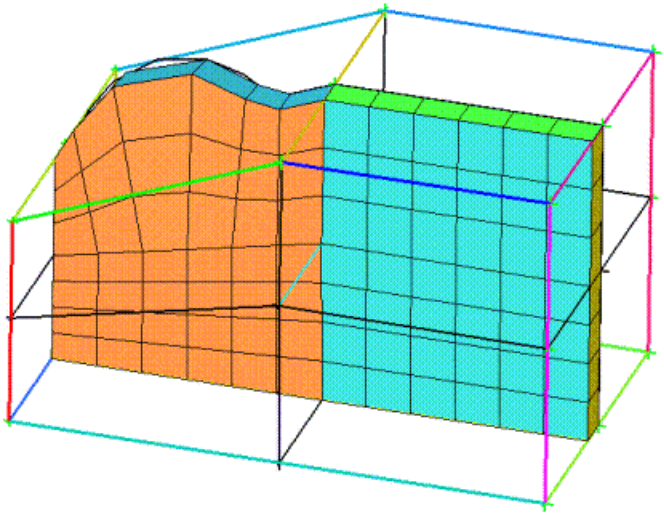
A model with two different materials is shown in Figure 372: Example of Link Shapes (p. 399). The **Shared Wall** option is set to **None**, so there is no projection between the parts. In figure (A), the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

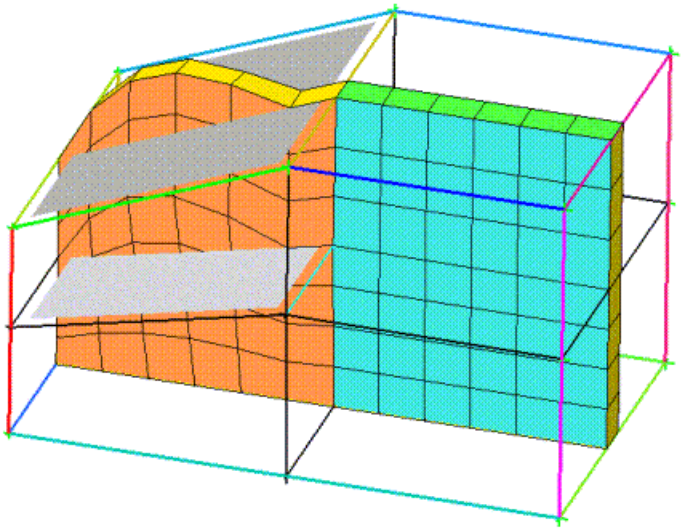
internal face is not linked to the top face. The top block projects to the curved surface (not shown), but the inner block is interpolated straight across the volume. After linking the internal face with the top face (figure (B)), the change in the mesh curvature transitions more gradually across both blocks instead of just the top one.

Figure 372: Example of Link Shapes

(A) Internal Face not Linked to the Boundary Face



(B) Internal Face Linked to the Boundary Face



Reference Mesh

allows you to use an existing surface mesh to seed an unstructured or swept face. This option can be used to obtain a better quality mesh or to set up node for node contact with a preexisting unstructured mesh.

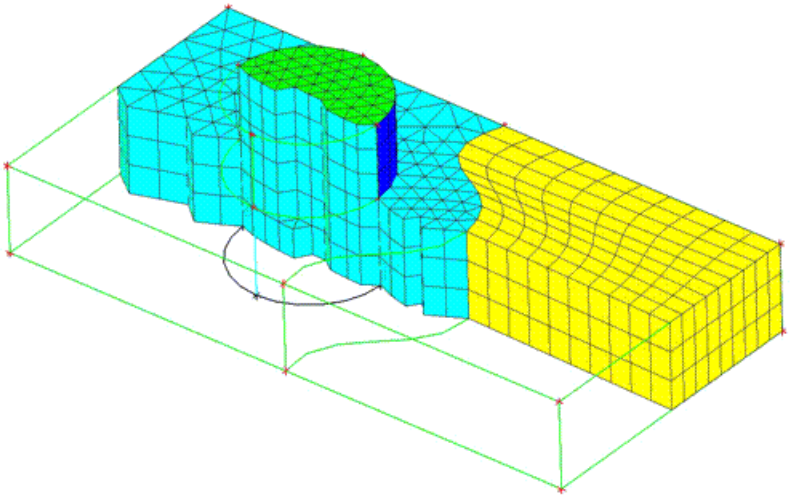
An example demonstrating the use of the **Reference Mesh** option is shown in Figure 373: Using the Reference Mesh Option (p. 400). The Multi-zone blocking on the geometry was generated automatically and one half mapped by default. The other half is an unstructured swept block with "all tri" free faces. To use a reference mesh, load an unstructured surface mesh and make sure the mesh perimeter matches the edge perimeter, including associations to surrounding curves. Verify the sweep direction (see the **Reverse Swept Direction** option) for the swept block. Select the **Reference**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

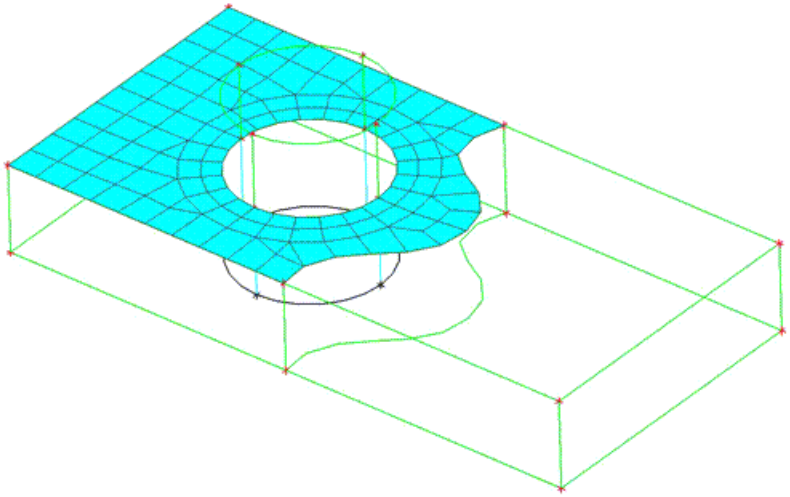
Mesh option, select the appropriate unstructured face and click **Apply**. The swept mesh will be seeded with the reference mesh.

Figure 373: Using the Reference Mesh Option

(A) The automatically generated Multi-zone blocking/mesh on the geometry.

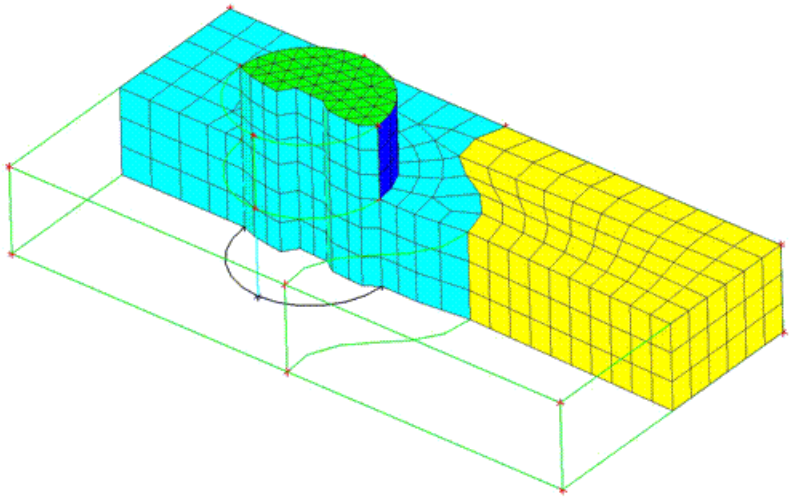


(B) The unstructured quad dominant surface mesh used as the reference mesh.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(C) The resulting swept mesh seeded with the reference mesh.



Selected Surface
allows you to associate the selected face(s) with the specified surface.

Disassociate from Geometry

The **Disassociate from Geometry** option allows you to disassociate the selected edge, surface or face that is associated to the geometry.

The selection of different entities to be disassociated can be done individually or together.

Note

Any association can be removed (associated to nothing), however, you can easily overwrite associations without disassociating first. For instance, if an edge is associated with a curve, you can directly associate it with a surface, you do not need to first disassociate it.

Update Associations

The **Update Associations** option allows you to set associations to the nearest entities of the assigned type.

Vertices
updates associations of all vertices.

Edges
updates associations of all edges.

Faces
updates associations of all faces.

Only Dormant entities
when enabled, will update associations for blocking that was associated with entities that are now dormant to the closest entities instead.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update Blocking

allows you to update blocking to a new geometry file. This option is used when there are geometric changes while the topology remains the same. **Update Blocking** uses the entity names. If blocking is created for one geometry file and saved, and then a second geometry file with consistent entity names is opened, the **Update Blocking** option will attempt to update the blocks to the new geometry file.

- **Parametric**

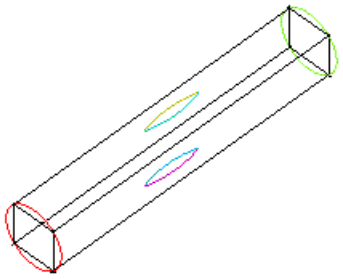
  When there is a parametric change in geometry, then Parametric method will update the blocking associations. When the initial blocking file is saved in relation to the first geometry file, the vertices contain links to the geometry in terms of the curve T parameter space and surface UV parameter space. Using the Parametric method will move the vertices to the same parametric space of the new geometry file.

- **Morphing**

  For some changes in geometry, such as geometric entities moved a large distance, or a Trim Surface operation, the Morphing Method should be used. This approach uses the Parametric method for curves, but for surfaces it will evaluate whether the surfaces are trimmed and adjust the vertices linked to the UV parameter space to a more accurate position based on attached curves. Instead of moving the surface vertices based on the UV parameter space, the surface vertices will be moved in reference to neighbor vertices.

The original geometry and blocking is shown in Figure 374: Original Geometry and Blocking (p. 402). The geometry is then scaled in the Z direction as shown in Figure 375: Geometry Scaled in Z Direction (p. 403). Finally, the result of updating the blocking associations is shown in Figure 376: Blocking Associations Updated (p. 403).

Figure 374: Original Geometry and Blocking



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 375: Geometry Scaled in Z Direction

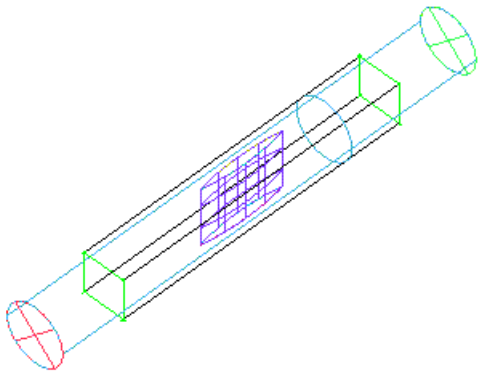
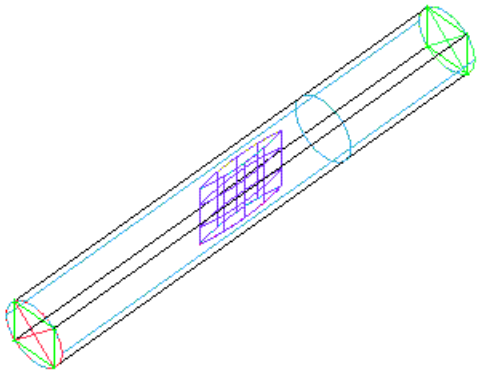
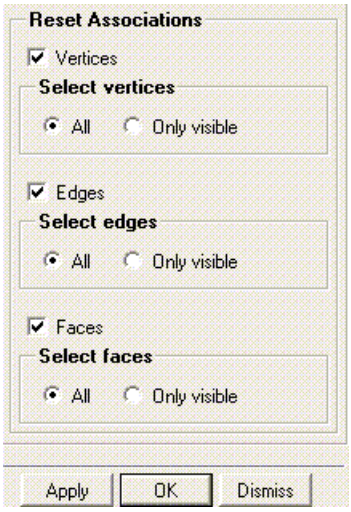


Figure 376: Blocking Associations Updated



Reset Associations

The **Reset Associations** option resets the associations of exterior blocking entities back to association with the nearest entity.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vertices
allows you to reset the vertex-point associations.

All
resets associations for all vertices.

Only visible
resets associations for visible vertices.

Edges
allows you to reset the edge-curve associations.

All
resets associations for all edges.

Only visible
resets associations for visible edges.

Faces
allows you to reset the face-surface associations.

All
resets associations for all faces.

Only visible
resets associations for visible faces.

Snap Project Vertices

The **Snap Project Vertices** option allows you to project all the vertices that are associated to respective points, curves or surfaces. The following options are available for projecting vertices:

All Visible
projects all visible vertices to their respective entity.

Selected
projects selected vertices to their respective entity.

Move O-Grid nodes
moves internal nodes (that are not projecting) which are attached via an O-grid edge to external nodes (nodes that are projecting) relative to the projection.

Note

Projection only applies to active parts. Projection will not be affected by entities that are blanked but are included in visible parts. If you do not want nodes to be moved to certain entities, the entities must be placed in a separate part that is disabled.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group/Ungroup Curves

The **Group/Ungroup Curves** option allows you to group curves into a composite curve, or ungroup composite curves into separate curves. This is needed for associating an edge to a set of curves. The curves should first be grouped into a composite curve, and then the edge can be associated to that composite curve.

Composite curves can be displayed by right clicking on Curves in the Display Tree and selecting the option **Show Composite**.

Group Curves
groups the selected curves into one composite curve. The selected curves can include existing composite curves.

Selected
groups only the selected curves.

All tangential
groups all curves that are tangential to the selected curves.

Part by Part
groups all the curves in each part into composite curves.

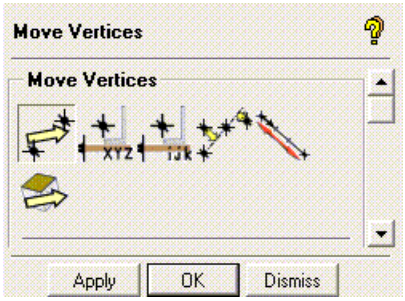
Ungroup Curves
allows you to select the composite curve to ungroup.

Auto Associate

The **Auto Associate** option allows you to associate edges to curves. Auto Association looks at the topology of the surfaces and the topology of the blocking and attempts to link the edge projections of the blocking in relation to the topology of the geometry.

Move Vertex

Figure 377: Move Vertices Options



The following options are available for moving vertices:
Move Vertex
Set Location

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Align Vertices
Align Vertices In-line
Set Edge Length
Move Face Vertices

Move Vertex

The **Move Vertex** option allows you to modify the location of a vertex. Select a vertex using the left mouse button, accept the selection by pressing the middle mouse button, and use the right mouse button to cancel the selection. After fixing the constraints, select the vertex to move.

Single Method
allows you to select a single vertex to be moved.

Multiple Method
allows you to select multiple vertices for movement.

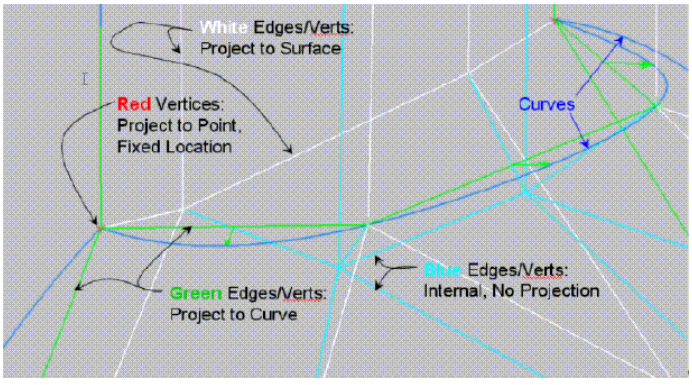
Movement Constraints
allows you to constrain a vertex to moving in any direction.

Normal to Surf
allows you to move the vertex normal to the surface.

Move dependent
allows you to move dependent vertices.

There are different types of vertices/edges, and their movement is governed by their projection as shown in Figure 378: Different Types of Vertices and Edges (p. 406).

Figure 378: Different Types of Vertices and Edges



White Edges/Vertices
These are between two material volumes. The edge and the associated vertices will be projected to the closest CAD surface between these material volumes. Vertices can only move on the surfaces.

Blue Edges/Vertices
These are in the volume. The blue vertices can be moved by selecting the edge just before it and can be dragged onto that edge.

Green Edges/Vertices
These edges and the associated vertices are being projected to curves. The vertices can only be moved on the curves that they are projected to.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Red Vertices
These cannot be moved, they are projected to fixed points. They can only be moved by first changing their projection to one of the other projection types.

Set Location

The **Set Location** option allows you to move vertices by setting their location with the following options:

Set Position
allows you to modify the coordinates of selected vertices. The vertices may be selected individually. You can also select a **Reference Point** and based on its position, modify the X, Y, and Z (or R, θ, and Z) directions of the selected vertices.

Multiple vertices can be selected and modified at the same time. Typically, this function is used to modify a number of vertices to one specific coordinate, for example Y=0, to ensure a true symmetry plane.

Reference From
allows you to select the reference point of the vertices to be moved. The reference point can either be an existing vertex, or a position on the screen.

Set Coordinate System
sets the appropriate coordinate system.

- Cartesian Coordinate System

  Select which direction to move the vertices: X, Y, Z, or the Normal direction. Enter the number of units of distance.

- Cylindrical Coordinate System

  Select which direction to move the vertices: R, θ, or Z direction. Enter the number of units of distance.

Vertices to Set
specifies the vertices to be moved. Select the vertices to be moved and click **Apply**.

Increment Position
allows you to move vertices by the specified increment in any direction. For example, vertices can be moved 10 units in the X-direction by entering "10" in the **Modify X** option.

Set Coordinate System
sets the appropriate coordinate system.

- Cartesian Coordinate System

  Select which direction to move the vertices: X, Y, Z, or the Normal directions. Enter the number of units of distance.

- Cylindrical Coordinate System

  Select which direction to move the vertices: R, θ, or Z directions. Enter the number of units of distance.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vertices to Set
specifies the vertices to be moved. Select the vertices to be moved and click **Apply**.

Align Vertices

The **Align Vertices** option allows you to align vertices along an edge to a selected reference vertex using planes. You can specify the plane in which the vertices will move.

Along edge direction
allows you to align all vertices along the selected edge to the reference vertex.

Reference vertex
specifies the vertex that the vertices along the selected edge will be aligned to. The vertex must be at a location along the edge.

Coordinate system
specifies the current active coordinate system, either Cartesian or Cylindrical.

- Cartesian Coordinate System

  Move in plane
  allows you to select the plane in which the vertices are to be moved. For the **User Defined** option, specify the vector normal to the desired plane.

- Cylindrical Coordinate System

  Parameter
  allows you to select the specified parameter of the vertices will be aligned to the reference vertex.

Align Vertices In-line

The **Align Vertices In-line** option allows you to align vertices to a defined line. Select two points to define the reference direction of the line, and then select the vertices to align to that line.

Set Edge Length

The **Set Edge Length** option allows you to modify the length of edges explicitly. Select the edge(s) to be modified and specify the length.

Freeze Vert(s)
allows you to override the default vertex adjustment and select which end vertex remains "frozen" in the same position when the edge length is modified. This option is disabled by default.

The default node movement is determined based on the association of the end vertices. If the association is the same, both vertices are moved equally. In other cases, the more restrictive vertex association is frozen automatically, and the other vertex is moved as per the length specified. Vertices associated to an entity will be restricted as opposed to non-associated vertices.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The order of restrictive association is association to point > association to curve > association to surface.

Note

When selecting multiple vertices to be frozen, ensure that only one of the end vertices of the edge being modified is selected.

Move Face Vertices

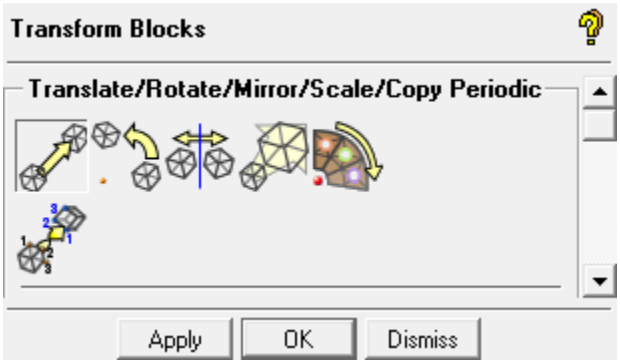
The **Move Face Vertices** option allows you to move or rotated face vertices.

Move Face Vertices
allows you to move the face vertices. Select the face whose vertices are to be moved. Specify the direction by entering an offset vector, or by selecting the start and end points for moving the face vertices.

Rotate Vertices
allows you to rotate the vertices. Select the vertices to be rotated, the center point, and the angle of rotation. Specify the rotation axis by entering a vector, or by selecting the start and end points.

Transform Blocks

Figure 379: Transform Block Window



The following options are available for transforming blocks.
Translate Blocks
Rotate Blocks
Mirror Blocks
Scale Blocks
Copy Periodic Blocking
Translate and Rotate

For the **Translate**, **Rotate**, **Mirror**, **Scale** and **Translate and Rotate** options, select the block(s) to be transformed. When no specific blocks are selected, it implies that the whole blocking will be transformed. You can specify whether to make copies of the blocks using the **Copy** option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the **Transform geometry also** option to transform the geometry only when transforming all the blocks. To use this option, do not explicitly select any blocks, which is functionally equivalent to selecting all blocks.

Translate Blocks

The **Translate Blocks** option allows you to move the original block topology laterally without rotation.

Select
Opens the **Select Blocking-block** menu to choose which blocks will be moved/copied.

Copy
If checked, the original block will remain, duplicate(s) will be created.

Transform geometry also
If checked, the geometry will be moved/copied with the selected block(s).

There are two methods to specify the distance of translation:

Explicit
allows you to specify the direction to move the block in X, Y, and Z directions.

Two Point Vector
allows you to specify the magnitude and direction to move the block by selecting two points to define a vector.

Rotate Blocks

The **Rotate Blocks** option allows you to rotate the original block topology.

Axis
specifies the axis of rotation. Select the X, Y, Z axes, or another user-defined vector.

Angle
specifies the value of the angle the block is to move.

Center of Rotation
specifies the center of rotation. Select the origin, centroid, or a user-defined point.

Mirror Blocks

The **Mirror Blocks** option allows you to mirror the topology about a specified plane.

The **Plane Axis** defines the plane orientation by its normal vector: X-axis, Y-axis, Z-axis, or a vector defined by two points.

Specify the **Point of Reflection** as the origin, centroid, or a user-defined point.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale Blocks

The **Scale Blocks** option allows you to change the size of the topology.

Scale Blocks by setting a factor for each direction of the coordinate system.

Specify the **Center of Transformation** as the origin, centroid, or a user-defined point.

Copy Periodic Blocking

The **Copy Periodic Blocking** option allows you to copy the blocking for periodic models. This feature is useful when the model is periodic but the geometry is available only for one sector. The blocking is first done on the sector geometry, and then both the geometry and blocking are copied and rotated to form the full model.

Note

The periodicity settings defined in Mesh > Set Up Periodicity will be applied.

Num. copies
specifies the number of copies of the geometry to be made.

Increment parts
allows you to select Parts that will remain the same and the copies will be placed in Parts with the same name plus an increment. By default, the copies of entities will be placed in the same Part as the original. For example, two copies of the Part “Blade” can be created and placed in Parts named “Blade_2”, and “Blade_3”.

Translate and Rotate

The **Translate and Rotate** option allows you to translate and rotate the blocking simultaneously. The reference location and target locations can be defined by 3 points, a curve or LCS and the blocking is translated and rotated to match.

Copy
If enabled, a copy of the selected entities will be created in the new location.

Translate and Rotate Method
The three options for method are as follows:

3 points –> 3 points
Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve
Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

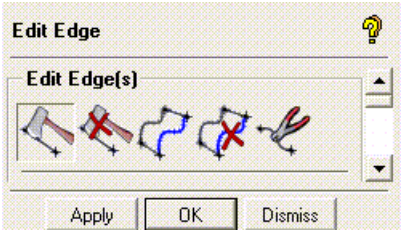
of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

LCS –> LCS
Click **Show LCS** to view any defined coordinate systems. Select two local coordinate systems.

The result will match the origins and align the axes of the first LCS to the second LCS.

Edit Edge

Figure 380: Edit Edge Options



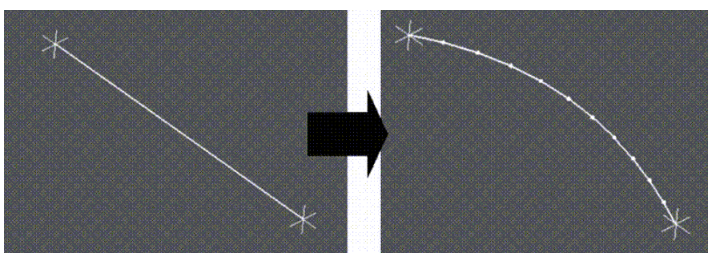
The following options are available for editing unstructured surface block edges:
Split Edge
Unsplit Edge
Link Edge
Unlink Edge
Change Edge Split Type

Split Edge

The **Split Edge** option allows you to edit block edges using the following **Split types**.

Spline
allows you to select the edge to be split and converted to a spline. Click on the edge and drag to specify the desired shape of the spline. The change will be applied when the mouse button is released. In Figure 381: Spline Type Edge Split (p. 412), the first picture shows the initial edge, and the second picture shows the edge after a spline split.

Figure 381: Spline Type Edge Split



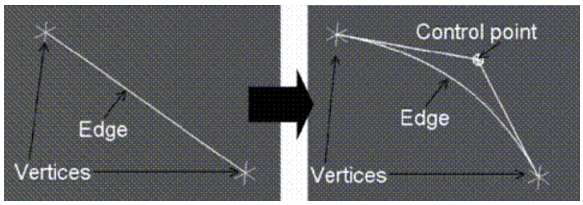
Linear
allows you to split the selected edge into linear edges. Figure 382: Linear Type Edge Split (p. 413) shows linear splits of a straight edge.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 382: Linear Type Edge Split

Control Point
is shown in Figure 383: Control Point Type Edge Split (p. 413).

Figure 383: Control Point Type Edge Split



Automatic Linear
allows you to automatically make the selected edge into a linear edge.

All Edges
allows you to convert all the edges into linear edges.

Note

This option corresponds to using the command **`ic_hex_auto_split_edge -all`** in the replay script.

Tangents
uses the tangencies of the attached edges at a vertex to a selected edge to split the selected edge. Each edge will be split twice, once for each vertex. For example, for a simple O-grid inside of a block, if the internal block's edges were split with this option, the internal block would take on a spherical shape.

Method
For each vertex, the edge can be split by one of the following methods:

Orthogonal
The length of the tangent edge will be based on the specified **Factor**. Enter a value or use the sliding bar.

Enter Tangent
Specify a **Tangency point** in relation to the vertex. The length of the tangent edge will be based on the specified **Factor**. Enter a value or use the sliding bar.

None
This option ignores the tangency of the vertex.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All at Vertex (Smooth)
allows you to split all the edges connected to the selected vertex to improve the angles between the edges at that vertex. If there are four edges, they will be split so that they meet at 90 degree angles at the vertex.

Tip

When using replay scripts, the command **ic_hex_split_edge** allows you to split the edge at the specified location.

For example, **`ic_hex_split_edge 21 25 0 1.7 8.5 0`** implies that edge 21–25 will be split at the location (1.7, 8.5, 0). The value **`0`** in this example indicates that this is the first split between the vertices 21 (start vertex) and 25 (end vertex). Alternatively, a value of **`1`** would indicate that the split is the second between the specified vertices, and so on.

To use an existing point to indicate the location to be used, you can use the following in the replay script:

```
set name pnt.00 ;# point must exist
set loc [ic_geo_get_point_location $name]
set x [lindex $loc 0]
set y [lindex $loc 1]
set z [lindex $loc 2]
ic_hex_split_edge 21 25 0 $x $y $z
```

Unsplit Edge

The **Unsplit Edge** option allows you to remove splits from edges. For the **Single** method, select the vertex of the split edge to remove. For the **All** method, select the edge to remove all the splits.

Link Edge

The **Link Edge** option allows you to set the shape of edges. Select the target edges and then the edge for the source of the shape.

Selected
sets the shape of only the selected edge.

Interactive
allows you to set the shape of the targeted edge(s) interactively.

In dimension
sets the shape automatically for all connected edges. Select the source edge for the **Link Edge Dimension**, the **Source index** or vertex, and the **Target index** or vertex.

Unlink Edge

The **Unlink Edge** option allows you to unset the shapes of edges linked by the **Link Edge** option.

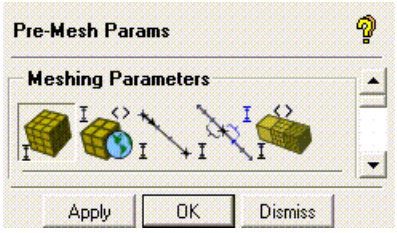
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change Edge Split Type

The **Change Edge Split Type** option allows you to change the edge split type (e.g., change spline to linear). The options available are **Spline**, **Linear**, and **Control Point**.

Pre-Mesh Params

Figure 384: Pre-Mesh Parameters Options



The following options are available for Pre-Mesh parameters:

Update Sizes
Scale Sizes
Edge Params
Match Edges
Refinement

Update Sizes

The **Update Sizes** option allows you to update sizes in the pre-mesh. The following options are available for updating sizes in the pre-mesh:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update All

computes the edge node spacing based on constraint equations with the default (BiGeometric) meshing law. You can adjust the number of nodes on each edge based on the **Global Surface** or **Curve Mesh Size** and by default each edge will follow the BiGeometric geometry law.

Note

The default meshing law and bunching ratio are specified under **Settings > Meshing > Hexa/Mixed**.

Note

You can check the distribution by clicking **Blocking > Edge > Show Edge Info** to verify that each edge follows the default (BiGeometric) geometry law.

Note

For 3D blocking, the **Curve Mesh Size** parameters are only applied to edges that are associated with curves. For 2D blocking, the **Curve Mesh Size** parameters are applied to all curves.

Keep Distributions

adjusts the number of nodes on each edge based on the **Global Surface** or **Curve Mesh Size**. It differs from the **Update All** method in that the edges follows the geometric law assigned to it, and not the default geometric law.

Note

For 3D blocking, the **Curve Mesh Size** parameters are only applied to edges that are associated with curves. For 2D blocking, the **Curve Mesh Size** parameters are applied to all curves.

Keep Counts

allows you to change the geometry law of the edges to the default (BiGeometric) geometry law. This option differs from **Update All** and **Keep Distribution** in that the distribution of nodes is not altered according to the **Global Surface** or **Curve Mesh Size**.

Curve->Edge bunching

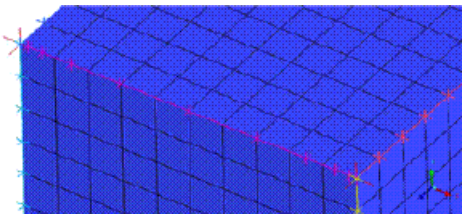
transfers the Advanced Bunching Parameters specified for curves to their associated edges.

You can apply Advanced Bunching to curves using the options available for **Curve Mesh Setup** in the **Mesh** tab. You can apply various bunching laws including Uniform, BiGeometric, and Hyperbolic, (described in the Bunching Laws (p. 422) section) along with end spacing, ratios and maximum spacing. This is primarily for use with the patch based surface meshers. **Pre-Mesh Params** uses the smallest average mesh size to calculate the initial pre-mesh distribution. The use of the **Curve -> Edge bunching** option to transfer the Curve Advanced Bunching Parameters to the associated edges is shown in Figure 385: Using the Curve->Edge Bunching Option (p. 417).

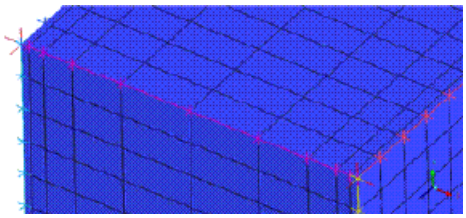
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 385: Using the Curve->Edge Bunching Option

(A) **Update all** does not respect Curve Advanced Bunching



(B) **Curve->Edge bunching** transfers Curve Advanced Bunching to Edge Parameters



In this case, only one curve had **Advanced Parameters** specified and this distribution was copied to parallel edges.

Note

Only intentionally set Advanced Bunching is transferred. Parallel edges without explicitly set Advanced Curve bunching will get the same distribution as their neighbors. This means it is not necessary to setup all the curves; parallel edges will automatically inherit the distribution from a single Curve with Advanced Bunching. To force the parallel distribution back to the default uniform bunching, you must select **Uniform** explicitly under **Curve Parameters** or adjust it afterward with the **Edge Parameters**.

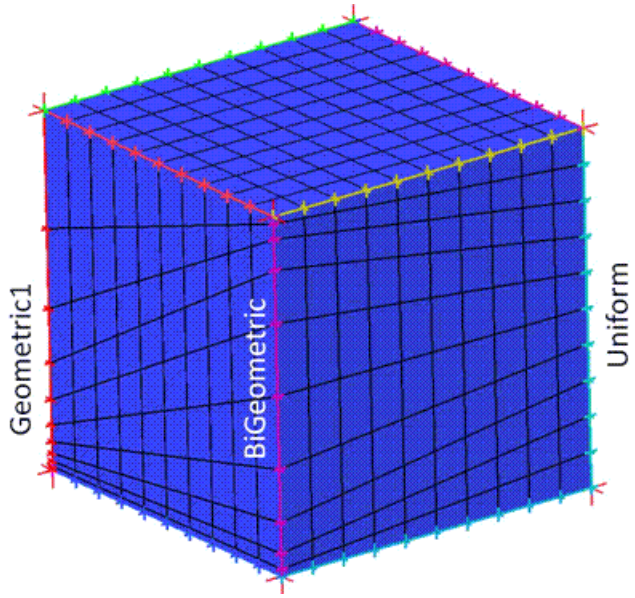
Note

This option is useful if you wish to store the Advanced Bunching with the Geometry file. Generally speaking, it is recommended that you use Blocking Edge parameters instead because there are more options and it is easier to copy or link parameters.

Multiple Curve Advance Bunching can be applied in parallel as shown in Figure 386: Multiple Curve Advanced Bunching (p. 418). Three different edge parameters are applied on the block shown. Edge distribution will be copied to parallel neighbors without explicit Curve Advanced Bunching. To make the bunching return to the default uniform bunching, Curve Advanced Bunching was explicitly set to **Uniform**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 386: Multiple Curve Advanced Bunching



Run Check/Fix Blocks

allows you to run the check for inconsistencies in the internal block data structures and fix them if possible. The default setting corresponds to that set for **Check/Fix Blocks** in the **Hexa/Mixed Meshing Options** DEZ.

Scale Sizes

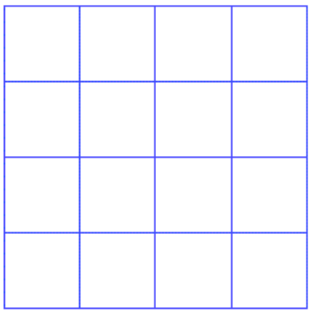
The **Scale Sizes** option allows you to scale the mesh size globally.

Factor

specifies the factor by which the mesh size will be globally multiplied.

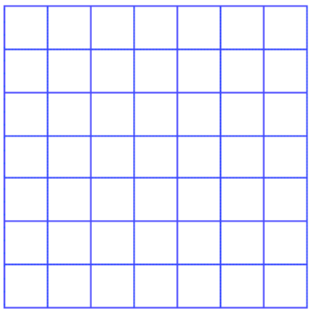
In Figure 388: Block Mesh Scaled by Factor of 1.5 (p. 419), the mesh size (from Figure 387: Initial Block Mesh (p. 418)) was changed by a factor of 1.5.

Figure 387: Initial Block Mesh



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 388: Block Mesh Scaled by Factor of 1.5



Scale Initial Spacings

allows you to adjust the end spacings on each of the edges and the max space (each will be divided by the factor) in addition to scaling the number of nodes. For example, if you entered a **Factor** of 1.5 and enabled **Scale Initial Spacings**, the number of nodes on each edge would increase by 50% while the end spacing on each edge would be reduced by 33%.

Note

This option only applies to the visible edges.

Edge Params

The **Edge Params** option allows you to modify the mesh parameters in a detailed manner by specifying various bunching laws and the node spacing along any particular edge. Each edge has several parameters that determine the spacing of the mesh along the edge: number of nodes, the meshing law, initial length at the beginning/end of the edge, the expansion of the mesh from the beginning/end of the edge to the interior, and the maximum element length along the edge.

The **Edge Params** icon brings up a window with all the mesh parameters. Once an edge has been selected, the mesh parameters for that edge will be displayed. All parameter values may be modified, except for the **Edge ID** and the **Edge Length**, which are pre-defined.

Note

You can also parameterize edge parameters using variables in replay scripts. Refer to Using Variables in the Replay Script for details.

Nodes

specifies the number of nodes along the edge. The number may be modified using the up and down arrows or by entering a number in the field.

Mesh law

Allows you to select one of several bunching laws described in detail in Bunching Laws (p. 422).

Spacing

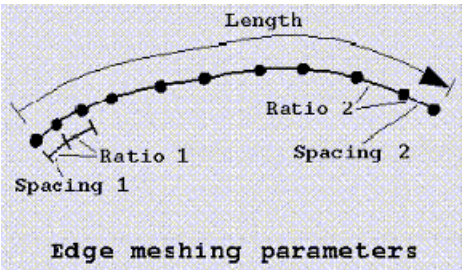
specifies the spacing of the first node from the beginning of the edge (first cell height). When an edge is selected, an arrow appears along the edge. **Spacing 1** refers to the parameters at the beginning end of the arrow, and **Spacing 2** refers to the edge end where the arrow is pointing, as shown in Fig-

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ure 389: Edge Meshing Parameters (p. 420). You may modify the values of the parameters available for the selected Meshing law. The actual values will differ from the requested values only if the requested values cannot be met. For example, if the Edge length is 10 units, and you specify an initial spacing of 6 on both sides and 11 nodes along the edge, the system will simply space the nodes evenly, giving an initial spacing of 1 and a spacing ratio of 1.

Ratio
is the growth rate from one cell height to the next. **Ratio 1** refers to the parameters at the beginning end of the arrow, and **Ratio 2** refers to the edge end where the arrow is pointing, as shown in the figure below.

Figure 389: Edge Meshing Parameters



Max Space
specifies the maximum element spacing of the curve.

Spacing Relative
if enabled, the values of **Spacing 1** and **Spacing 2** are displayed as fractions of the edge length.

Nodes Locked
if enabled, the number of nodes will be fixed. However, **Update All** will override this and apply the global parameter values to the mesh.

Parameters locked
if enabled, the Mesh law parameters will be fixed. However, **Update All** will override this and apply the global parameter values to the mesh.

Copy Parameters
allows the bunching on the currently selected edge to be copied with various options. If the **To All Parallel Edges** option is selected, the distribution of the current edge will be copied to all parallel edges. Parallel edges are edges that start and end with the same index value. If the **To Visible Parallel Edges** option is highlighted, the distribution of the current edge will be copied to just the visible parallel edges. By default, the bunching parameters of the parallel edges are based upon the relative lengths of the selected edge and destination edges. The **To Selected Edges** option allows you to copy edge parameters to selected edges. The **From Edge** option allows you to copy a node distribution from an edge to the currently selected edge. You will be prompted to select the edge to copy the node distribution from. The **Reverse** options will copy the node distribution and reverse it during the copy operation.

Copy Absolute
if enabled, the exact spacing from one edge will be copied to the specified edges, regardless of edge length.

Linked bunching
allows the distribution of nodes on a single edge to be identical to the distribution of nodes on a series of smaller parallel edges. Linked bunching allows you to define a permanent relationship, called links,

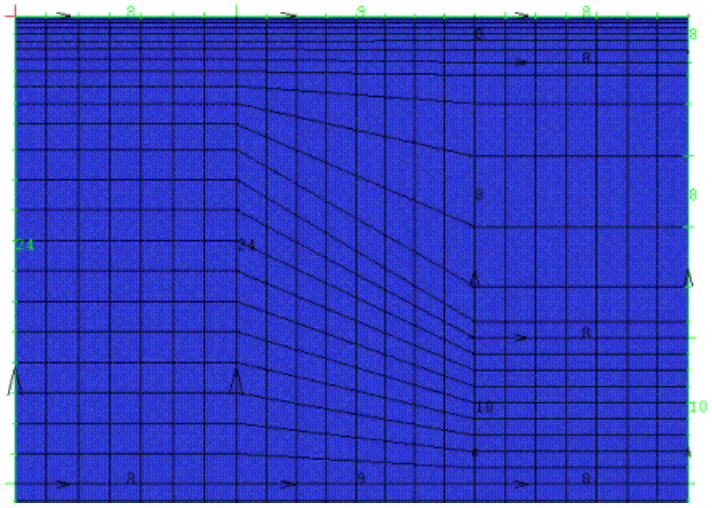
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

between these edges. For example, if the node distribution must be modified on the smaller edges, the user does not have to specify any node distribution on the larger edge. The node distribution on the larger edge will automatically be updated to reflect the node distribution on the smaller edges. This is primarily used if there are splits that do not cross a block, but mesh distribution on both sides need to match, as shown in the example below.

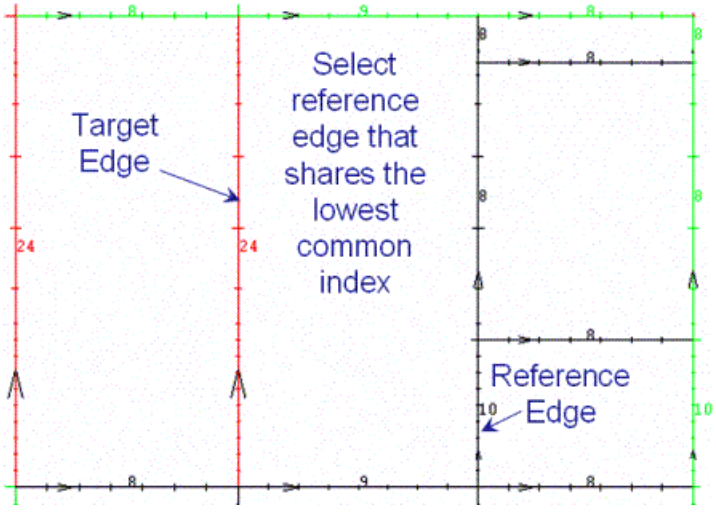
To use this option, first click the select icon for the **Edge** field (in the **Pre-Mesh Params > Edge Params** DEZ and select the target edge, which is the longer edge with more nodes. Enable **Linked bunching** and click the select icon to select the reference edge. It is important to select the parallel edge with the same base index as the target edge. In the example in Figure 390: Linked Bunching Example (p. 421), the lowest of the three edges is selected because it shares the same base index with the target edge.

Figure 390: Linked Bunching Example

Example of parallel edges with different node distributions

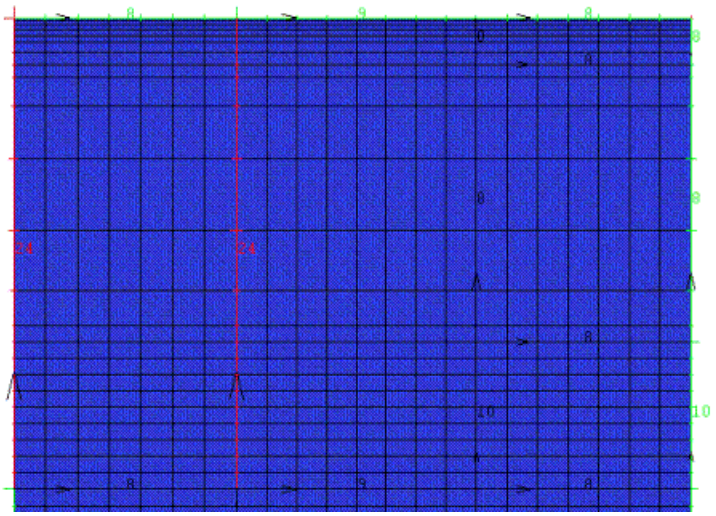


Target and Reference Edge selection



Edges after Linked Bunching option is applied

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Highlight dependent edges
highlights in red the dependent edges of the edge selected.

Highlight attached faces
highlights the faces attached to the selected edge.

Highlight master edges
highlights in yellow the linked master edge of the selected edge. Linked edges depend on the master edge for their parameters.

Reverse parameters
allows you to reverse the parameters for the ends of the currently selected edge. The parameters at the beginning of the edge (end 1) are switched with the parameters at the end (end 2).

Note

The number of nodes and the meshing laws specified in the **Edge Mesh Parameters** always take precedence in determining the number of nodes. Then spacings 1 and 2 are equally balanced, followed by ratios 1 and 2, which are also equally balanced, and finally Max Space is considered.

Bunching Laws

BiGeometric
The default bunching law. The two initial heights and ratios define parabolas in a coordinate system where the number of node points is the X-axis and the cumulative distance along the edge is the Y-axis. The parabolas are truncated where their tangent lines are identical; the spacing is linear between these points. If there are not enough nodal points to form this linear segment, a hyperbolic law is used and the ratios are ignored.

Uniform
The nodes along the edge are uniformly distributed.

Hyperbolic
The spacing at each end are used to define a hyperbolic distribution of the nodes along the edge. You can set Spacing 1 and Spacing 2, and the growth ratios are determined internally.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Hyperbolic Tangential Bunching law is described by the following equations:

$$S_i = \frac{U_i}{2 \cdot A + (1 - A) \cdot U_i}$$

where:

$$U_i = 1 + \frac{\tanh\left(b \cdot R_i\right)}{\tanh\left(\frac{b}{2}\right)}$$

$$R_i = \frac{i-1}{N-1} - \frac{1}{2}$$

$$A = \sqrt{\frac{Sp1}{Sp2}}$$

$$\sinh\ b = \frac{b}{(N-1) \cdot \sqrt{Sp1 \cdot Sp2}}$$

The parameter limitations are:

$$0.000001 < \mathrm{Sp1} < 0.000000$$

$$0.000001 < \mathrm{Sp2} < \min\left(\frac{0.999999}{(N-1)^2 \cdot Sp1}; 0.999999\right)$$

Poisson

The spacing of the node intervals is calculated according to a Poisson distribution. Requested values of Spacing 1 and Spacing 2 are used. Requested values of Ratio 1 and Ratio 2 are not used directly, but are used to determine if the requested spacing is appropriate. If not, the spacing will be adjusted automatically.

The mapping function is obtained by solving the following differential equation:

$$\frac{d^2}{dt^2} x_0(t) + P(t) \cdot \frac{d}{dt} x_0(t) = 0 \quad ,$$

with the following boundary conditions:

$$\begin{aligned} &x_0(1) = 0 \\ &x_0(2) = S_2 = Sp1 \\ &x_0(N-1) = S_{N-1} = 1 - Sp2 \\ &x_0(N) = S_N = 1 \end{aligned} \quad ,$$

where Sp1 = Spacing 1 and Sp2 = Spacing 2.

The function P is required to satisfy the Neumann boundary condition. It is computed by an iterative optimization process loop. Some parameter limitations are:

0.0 < Sp1 < 1.0

0.0 < Sp2 < 1.0 - Sp2

500 < Number of iterations < 9999

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curvature

The spacing of the node intervals is calculated according to the curvature of the function defining the distribution.

Geometric 1

Spacing 1 is used to set the first distance from the starting end of the edge, with the remaining nodes are spaced with a constant growth ratio. Only Spacing 1 is specified.

The Geometric bunching laws are described by the following equation: $S_i = \frac{R-1}{R^{N-1}-1}\sum_{j=2}^{i} R^{j-2}$,

where S_i is the distance from the starting end to node *i*, R is the ratio, and N is the total number of nodes. The ratio R is limited by 0.25 > R > 4.0.

Geometric 2

The same as Geometric 1, except that Spacing 2 is used to define the distribution starting from the terminating end of the edge.

Exponential 1

The Exponential 1 bunching law is described by the following equation: $S_i = Sp1 \cdot i \cdot e^{R(i-1)}$

where S_i is the distance from the starting end to node *i*, Sp1 is Spacing 1, N is the total number of nodes, and R is the ratio, where: $R = \frac{-\log\left[(N-1)\cdot Sp1\right]}{N-2}$.

Exponential 2

The same as Exponential 1, except that the Spacing 2 and Ratio 2 parameters are used and the distribution starting point is the terminating end of the edge.

Biexponential

The spacing of the node intervals is calculated according to the law specified for Exponential 1 and 2. However, Spacing 1, Ratio 1, Spacing 2 , and Ratio 2 are used to define the distribution. The Spacing 1 and Ratio 1 parameters define the distribution from the beginning of the edge to the midpoint of the edge, and Spacing 2 and Ratio 2 define the distribution from the terminating end of the edge to the midpoint of the edge.

This bunching law is described by the following equation: $S_i = \int_0^i Exp\left(a_0 + a_1 x + a_2 x^2 + a_3 x^3 + a_4 x^4\right) dx$

The parameters are computed according to the vertex constraints. If a ratio equals 0 at a vertex, the spacing constraint at this vertex only is taken into account and the ratio constraint with the neighbor spacing is left free by decreasing the polynomial order in the mathematical function.

Linear

The spacing of the node intervals is calculated using a linear function.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Match Edges

The **Match Edges** option allows you to match the edge spacing of a reference edge to a connecting target edge. The node spacing on the end of the target edge that connects to the reference edge will be modified to match the node spacing on the reference edge. The following methods are available:

Selected
allows you to match the end spacing of the selected target edge(s) to the adjacent end spacing of the reference edge. Matching is done around a connected node, so with the interactive **Selected** method, all the target edges selected must be connected to the same node as the reference edge. Match edges is a quick way to ease edge bunching size transitions between connected edges and adjacent blocks.

Link spacing
links the target edge end spacing with the spacing on the reference edge. Further adjustments to the reference edge end spacing will automatically be carried to the linked target edges making updates easier. To remove or modify linked spacing, use the **Edge Parameters** DEZ. Alternatively, a match is reset when a new match is specified around the same vertex, with or without the link spacing.

Automatic
allows you to automatically match edge spacing at selected vertices.

Vertices
specifies the selected vertices.

Spacing

Minimum
selects the edge having the minimum spacing defined at the selected vertex as the reference edge.

Maximum
selects the edge having the maximum spacing at the selected vertex as the reference edge.

Average
averages the spacing at the selected vertex.

Match Edges Dimension
allows you to select the dimension to be matched. For example, you may want to match edges only in the O-grid direction (o3). Or, for a turbine blade model, you may match in the I and J directions first to match the edges in the plane, and then match in the Z direction for even distribution perpendicular to the plane. You can use the **Select** option and select the reference edges to determine the index directions for matching, or use the **All** option to match the edges in all directions at once.

Copy bunching
allows you to copy the edge bunching from the selected reference edge to either selected edges or all parallel edges. When used with scripting, this function has the advantage of listing the “from edge” and “to edge” rather than the specific edge parameters that existed during the recording. This is more parametric than copying bunching from the **Edge Parameters** DEZ.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

Selected Edges
allows you to copy the bunching from the reference edge to the selected target edge(s). This copies the bunching law, end spacings, end ratios and number of nodes.

You can choose to **Exclude Number of Nodes** if you do not want that parameter copied.

All Parallel Edges
allows you to copy the bunching from the reference edge to all parallel edges. If **Copy absolute** is enabled, the exact spacing from one edge will be copied to the specified edges, regardless of edge length.

Refinement

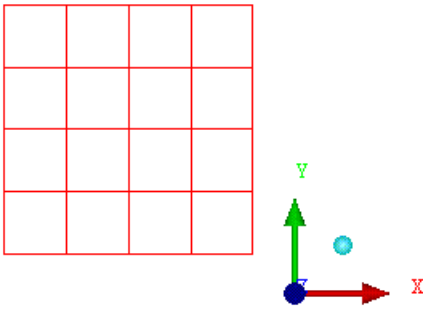
The **Refinement** option allows you to refine blocks by a scale factor, as shown in Figure 393: Block Refined (p. 427). A **Level** value greater than 1 will result in refinement, while a **Level** value less than 1 will result in coarsening of the pre-mesh.

Blocks
allows you to select the blocks to be refined.

Level
specifies the number of levels to refine the blocking.

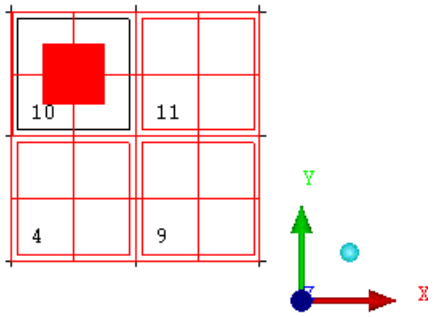
Refinement Dimension
If **All** is selected, the blocks will be refined in every dimension. If **Selected** is chosen, you can enter the dimension (edge) that specifies the direction in which to modify the block. The value 0 corresponds to the X or I direction, 1 corresponds to the Y or J direction and 2 corresponds to the Z or K direction.

Figure 391: Initial Mesh



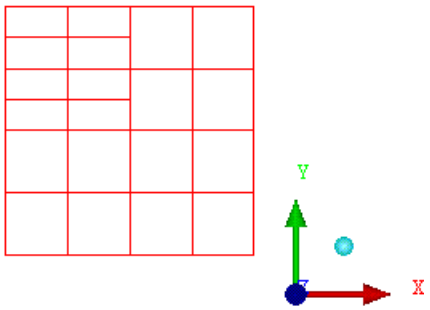
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 392: Selection of Blocks and Edge for Refinement



The result is shown below, in which the block is refined in the selected region and along the selected dimension.

Figure 393: Block Refined



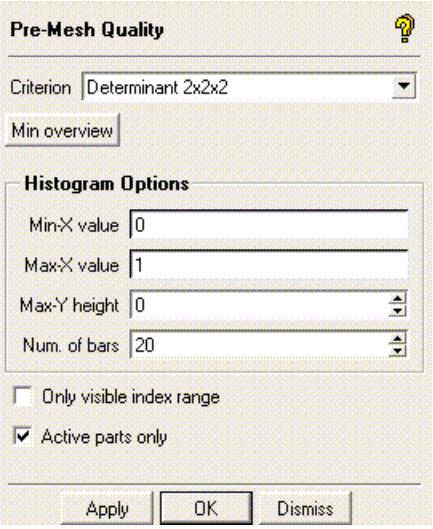
Pre-Mesh Quality

The **Pre-Mesh Quality** option allows you to check the mesh quality. All the elements of the current volume family will be included in the quality check.

Note

Quality for volume elements only is calculated for 3D blocking. If you use Edit Mesh > Display Mesh Quality for the same mesh, the quality may be different if surface elements are included for that calculation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Criterion
specifies the quality criterion. The criteria available for selection are described in Pre Mesh Quality Options (p. 428).

Min overview
reports the minimum quality for all applicable metrics in the message window.

Histogram Options
allows you to set the minimum and maximum X-axis values, the maximum Y-height, and the number of bars displayed in the histogram.

Note

You can also change the histogram display settings and ranges by right-clicking in the histogram window.

Only visible index range
if the visible index range is reduced, then only the elements that are in the index range set by the **Index Control** will be checked. The Index Control window is accessible from the menu that appears when right clicking on **Blocking** in the **Display Tree**.

Active parts only
if enabled, only the elements within active parts (Display Tree) will be checked.

Pre Mesh Quality Options

The quality can be checked using many different criteria, as described in the following sections:

Angle
Aspect Ratio
Constant Radius
Custom Quality
Determinant (2x2x2 stencil)
Determinant (3x3x3 stencil)
Distortion
Equiangle Skewness
Eriksson Skewness

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ford
Hex. Face Aspect Ratio
Hex. Face Distortion
Max Angle
Max Dihedral Angle
Max Length
Max Ortho
Max Ortho 4.3v
Max Ratio
Max Sector Volume
Max Side
Max Warp
Max Warp 4.3v
Mid Node
Mid Node Angle
Min Angle
Min Ortho
Min Sector Volume
Min Side
Opp Face Area Ratio
Opp. Face Parallelism
Orientation
Quality
Taper
Volume
Volume Change
Warpage
X Size
Y Size
Z Size

Angle

This option checks the minimum internal angle for each element. The default range is 0–90 degrees, with 0 as degenerate and 90 as perfect.

Aspect Ratio

For hexahedral elements, the **Aspect ratio** is defined as the size of the minimum element edge divided by the size of the maximum element edge. The values are scaled and the default range of values is 1–20, such that an **Aspect ratio** of 1 indicates a regular element.

Constant Radius

This option computes the distortion of a plane, based upon the nodes that compose the surface. The less warped the plane is, the better quality of mesh the model will have. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Custom Quality

This option allows you to define your own quality metric as the combination of the following quality criteria and element types: Determinant, Warp, Min Angle, and Max Angle.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify the maximum and minimum quality levels of a histogram for quad and tri elements. All values are adjusted to a scale of 0 to 1 when **Recompute** is pressed.

For example, if the Min Angle Quad is 30 degrees and the maximum is 45 degrees, then 30 degrees will correspond to the minimum value in the custom quality and 45 degrees will correspond to the maximum value.

Determinant (2x2x2 stencil)

The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix. In this option, the determinant at each corner of the hexahedron is found. The default range is 0–1 with a Determinant value of 1 indicating a perfectly regular mesh element and 0 indicating an element degenerate in one or more edges. Negative values indicate inverted elements.

Note

This determinant is appropriate for 3D linear hexas and calculates more quickly than the 3x3x3 matrix because it does not include the unnecessary mid side nodes.

Determinant (3x3x3 stencil)

This is for hexahedral elements. This option is the same as the 2x2x2 stencil, but edge midpoints of blocks are added to the Jacobian computation.

The Jacobian determinants for hexahedras will be calculated at r,s,t = -1,0,1 of the natural coordinate system of the element (27 node positions). Next it calculates the maximum absolute determinant of the 27 determinants (3x3x3). If this is at position i with absolute determinant value max0, then for each of the 27 positions (j) (except i) the absolute distance of determinant j to determinant i will be calculated. The final result will then be 1 minus the maximum of the absolute distances divided by max0, so that the range of this quality criterion value will be between -1 and 1. The Jacobian determinant is the determinant of the Jacobian operator which connects the derivatives of the natural coordinates (r,s,t) with the derivatives of the local coordinates (x,y,z). J = ((dx/dr dy/dr dz/dr) (dx/ds dy/ds dz/ds) (dx/dt dy/dt dz/dt)).

Note

A good book to understand the determinant calculation is: Finite Element Procedures, by K.J. Bathe, Prentice Hall, New Jersey 07632, 1996.

Distortion

This is available only for hexa elements and is the measurement of the twisting of the element from the ideal shape. For this the Jacobian determinants will be calculated at r,s,t=-1,0,1 of the natural coordinate system of the element (27 node positions). The distortion is 27 times the minimum of all absolute determinants divided by the sum of all 27 absolute determinants (0 if all Jacobian determinants are 0).

Equiangle Skewness

This quality parameter applies to tetra, hexa, quad, and tri elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element equiangle skew = 1.0 – max ((Q_{max} – Q_e) / (180 – Q_e), (Q_e – Q_{min}) / Q_e),

where

Q_{max} = largest angle in the face or element

Q_{min} = smallest angle in the face or element

Q_e = angle of an equiangular face or element (e.g., 60 degrees for a triangle, and 90 degrees for a square).

Eriksson Skewness

This is an empirical criterion, obtained for a hexahedral element by dividing the volume of the closest parallelepiped by the product of its edges. It measures the shear of the parallelepiped closest to the current element using least squares approximation. The default range of values is 0–1. Generally acceptable elements have skewness ranging from 0.5 to 1.

Ford

This is a hybrid quality parameter for 3 and 4 node (tri and quad shell) element meshes based on weighted skewness, warpage, and aspect ratio values. It was custom developed for Ford Motor Company to fit their established processes and metrics. The possible range is from 0 to 32.

Hex. Face Aspect Ratio

This quality parameter calculates the 3 averaged face areas (average of the areas of two opposite faces) of hexahedra elements. The maximum of the six possible divisions of the averaged face areas will be calculated and then inverted to normalize the result.

Hex. Face Distortion

This quality parameter calculates the product of the maximum edge size in the i direction multiplied by maximum edge size in the j direction, multiplied by the same in the k direction, and then divides this value by the volume of the hex element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Angle

This calculates the maximum internal angle of each element. The range of values is 90–180 degrees.

Max Dihedral Angle

This is the maximum angular space contained between planes which intersect. It is measured by the angle made by any two lines at right angles to the two planes. The range of values is 90–180 degrees.

Max Length

This calculates the maximum length of the diagonals of quad faces, and the maximum side length of tri faces. This works for all meshes and element types. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Ortho

This calculates the maximum deviation of the internal angles from 90 degrees for each element. The default range is 0–90.

Max Ortho 4.3v

This calculates the maximum deviation of the internal angles of the elements from 90 degrees. For elements other than hexas this diagnostic is equal to Max Ortho. For hexas this differs from Max Ortho in the way that angles between 180 and 360 degrees are also considered (deviation up to 270 degrees).

Max Ratio

This calculates the maximum ratio of the lengths of any two edges that are adjacent to a vertex in an element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the maximum of the calculated sector volumes will be taken. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Side

This calculates the maximum side length of each element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Max Warp

This calculates the maximum warp (in degrees) of all elements. This works only for structured volume and surface meshes, linear hexahedral and linear quadrahedral elements. To determine warp of a quadrilateral face, find the midpoints of all edges, which will be co-planar. Then, calculate the maximum angle of any edge with the plane thus defined, which is the warp of the face. The maximum warp of a volume element or hexahedron is the maximum warp of its faces. For a 2D, planar mesh of **QUAD_4** elements, the warp will be zero for all elements. The default range is 0–90 degrees.

Max Warp 4.3v

This measure applies to quad, prism and hexa elements (for quadratic elements the linear part will be checked).

To determine the warp of a quadrilateral face, the angles between the triangles connected at the 2 diagonals of the quad will be calculated and the maximum will be used.

The maximum warp of a volume element (prism or hexa) is the maximum warp of its quad faces.

For a 2D planar mesh of QUAD_4 elements, the warp will be zero for all elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

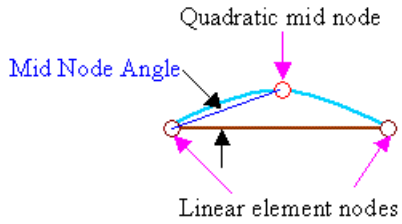
Mid Node

This calculates the maximum deviation of the mid node. The default range is 0–1.

Mid Node Angle

This calculates the angle by which the quadratic mid node is off from the linear edge. The default range is 0–90.

Figure 394: Definition of Mid Node Angle



Min Angle

This calculates the minimum internal angle of each element. The default range is 0–90 degrees.

Min Ortho

This calculates the minimum deviation of the internal angles from 90 degrees for each element. The default range is 0–90.

Min Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the minimum of the calculated sector volumes will be taken. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Min Side

This calculates the minimum side length of each element. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Note

Use this check if you are concerned about Min Side reducing the time step. There is also a modified Laplace smoother that can help increase these min side elements.

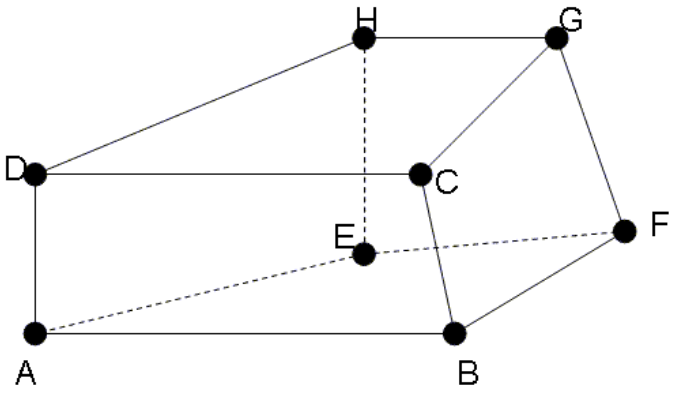
Opp Face Area Ratio

This is applicable for hexahedral elements only. This is the measurement of the worst ratio of the areas of the opposite faces of the hexahedral element. Ideally, this value should be 1. The default range is

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Figure 395: Definition of Opposite Face Area Ratio



In the figure above, let A1 = Area of Quad Face {ABDC}, and A2 = Area of Quad Face {EFHG}. If A1>A2, then the Opposite Face Area Ratio of this pair of faces = A1/A2. If A1<A2, then the Opposite Face Area Ratio = A2/A1. Similarly, the Opposite Face Area Ratio is found for each opposing pair of faces, and the maximum of all three pairs is found as the measurement for this hexahedral element.

Opp. Face Parallelism

This feature is also for hexahedral elements only. It is the measurement of the parallelism of the hexahedral elements. If the opposite faces are ideally parallel, the value is 1. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Orientation

This computes the direction of the face normal determined by the orientation of the nodes based on the right hand rule. Face orientation should be into the volumetric domain.

Quality

The criterion **Quality** is calculated differently for different element types.

- **Tri and Tetra**

  The quality is calculated as the minimum ratio of height to base length of each side (normalized to 1).

- **Quad**

  The quality is calculated as the Determinant, as described in Determinant (2x2x2 stencil) (p. 430).

- **Hexa**

  The quality is a weighted diagnostic between Determinant (between -1 and 1), Max Orthogls (normalized between -1 and 1; if deviation from orthogonality is greater than 90 degrees, then the normalized value will be smaller than 0) and Max Warpgls (normalized between 0 and 1;

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

warpage of 0 degrees is 1, warpage of 180 degrees is 0). The minimum of the 3 normalized diagnostics will be used.

- **Pyramid**

  The quality is calculated as the determinant.

- **Prism**

  The quality is calculated as the minimum of the Determinant and Warpage. Warpage is normalized to a factor between 0 and 1, where 90 degrees is 0, and 0 degrees is 1.

Taper

For hexahedral elements, the **Taper** is the maximum ratio of the areas of opposite faces. For quad elements, it is the maximum ratio of the lengths of opposite edges.

Volume

This computes the volume of each element based on the corner points. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Volume Change

This quality metric is calculated for a specific element by finding the maximum volume of all its neighboring elements and dividing it by the volume of the element itself. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Warpage

This computes the distortion of a plane, based upon the nodes that compose the surface. The default range is 0–90, where a warpage value of 0 is flat (preferred) while 90 is degenerate.

X Size

This computes the length of the element profile in the X direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Y Size

This computes the length of the element profile in the Y direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

Z Size

This computes the length of the element profile in the Z direction. The default range is 0–20, but if the computed range is above 20, the **Min Value** will be set to zero and the **Max Value** will be automatically adjusted to ensure the histogram includes all existing elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

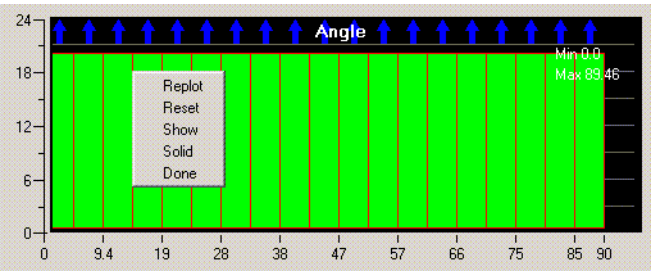
Pre-Mesh Quality Histogram

After computing the determinant values of the elements, a histogram of the values will be displayed. The X-axis represents the quality range for the particular type, and the Y-axis represents the number of elements within a particular histogram bar. An arrow at the top of a histogram bar indicates that there are more elements in that bar than are displayed per the current maximum number on the Y-axis.

By clicking on any of the bars with the left mouse button, the information about the precise number of values that fall into this interval and its boundaries are displayed in the Messages window. Bars that are clicked on change color, and remain selected until clicked on again. To display the elements within a selected (highlighted) histogram bar, right click on the histogram window, and select **Show**.

The histogram of the Angle Quality for a certain hexahedral block is shown in Figure 396: Histogram of Angle Quality (p. 436). The histogram options menu is also shown.

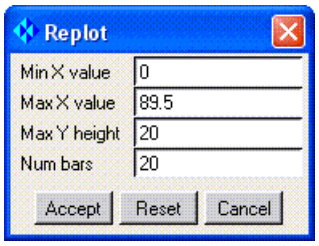
Figure 396: Histogram of Angle Quality



The histogram options are as follows:

Replot
allows you to change the parameters of the histogram using the **Replot** window.

Figure 397: Replot Window



Min / Max X value
sets the minimum and maximum values of the particular quality type selected for the X-axis.

Max Y height
sets the maximum number of blocks for the quality ranges. A value of 0 will set the display to such that the histogram with the largest number of elements is fully visible. A smaller number will give increased resolution for quality ranges with fewer elements.

Num bars
sets the number of bars to use in the histogram display.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset
sets the maximum number on the Y-axis such that the histogram bar with the largest number of elements is fully visible.

Show
displays the elements within the selected (highlighted) histogram bars.

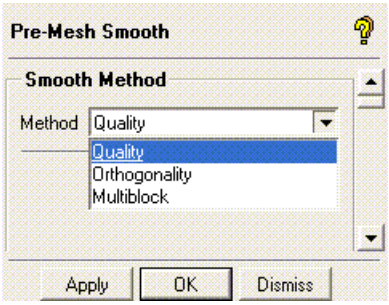
Solid
redraws all selected elements using flat shading, according to the color map.

Done
closes the histogram window.

Pre-Mesh Smooth

The **Pre-Mesh Smooth** option allows you to smooth the pre-mesh.

Figure 398: Pre-Mesh Smooth Options



There are three options available for smoothing blocking mesh as shown in Figure 398: Pre-Mesh Smooth Options (p. 437).
Quality Method
Orthogonality Method
Multiblock Method

Quality Method

The following options are available for the Quality method for pre-mesh smoothing:

Smoothing iterations
specifies the number of smoothing iterations to be run.

Up to quality
specifies the quality level for smoothing. The default is 0.2.

Criterion
specifies the quality criterion for smoothing.

Determinant 3x3x3
Smoothing by Determinant 3x3x3 tries to locally improve the worst determinant of all elements hanging on a node.

Angle
Smoothing by Angle tries to locally improve the worst angle of all elements hanging on a node.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Options

contains advanced options for smoothing.

Only visible index range

If enabled, only the elements that are in the index range set by the **Index Control** will be checked. The Index Control window is accessed from the menu that appears when right clicking on **Blocking** in the **Display Tree**.

Active parts only

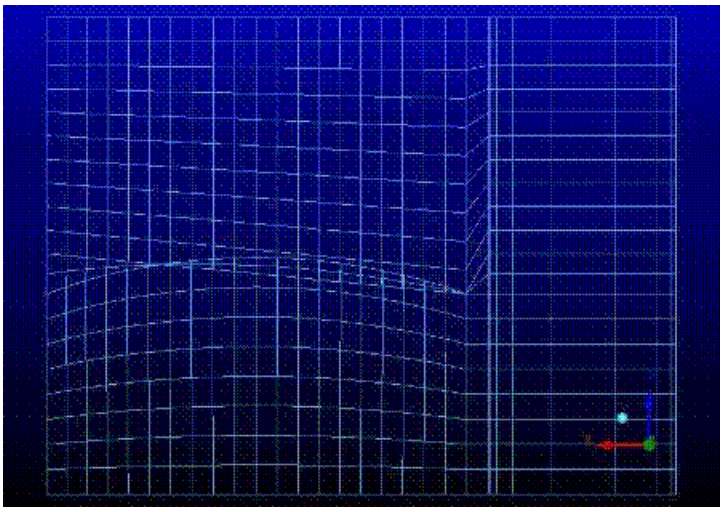
If enabled, only the active parts will be smoothed.

Laplace smoothing

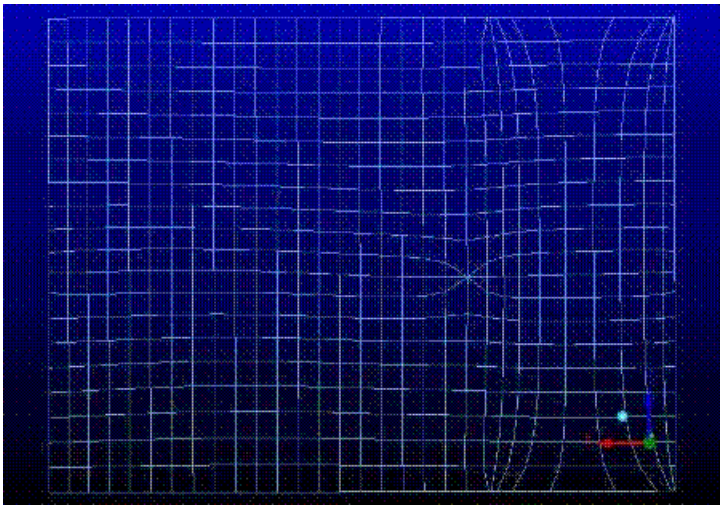
If enabled, performs Laplace smoothing for the surface. Laplace smoothing works on the whole mesh and tries to create uniform transition.

Figure 399: Example of Laplace Smoothing

Original Mesh



Mesh after Laplace smoothing



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orthogonality Method

The Orthogonality method relaxes unstructured hexahedral meshes in order to obtain smooth grid lines orthogonal to the boundary as well as smooth grid angles and transitions in the inner volume. It first smoothes the surface mesh recognizing the topological boundary edges. If the number of volume smoothing steps is greater than 0, after each surface smoothing step the inner volume will be adjusted by performing 1 volume smoothing step. After the surface smoothing has been finished, the inner volume will be smoothed (according to the number of volume steps set). It can also smooth real surface meshes.

Note

A minimum of 10 iterations of smoothing of the surface mesh must be completed for stability reasons.

The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

Note

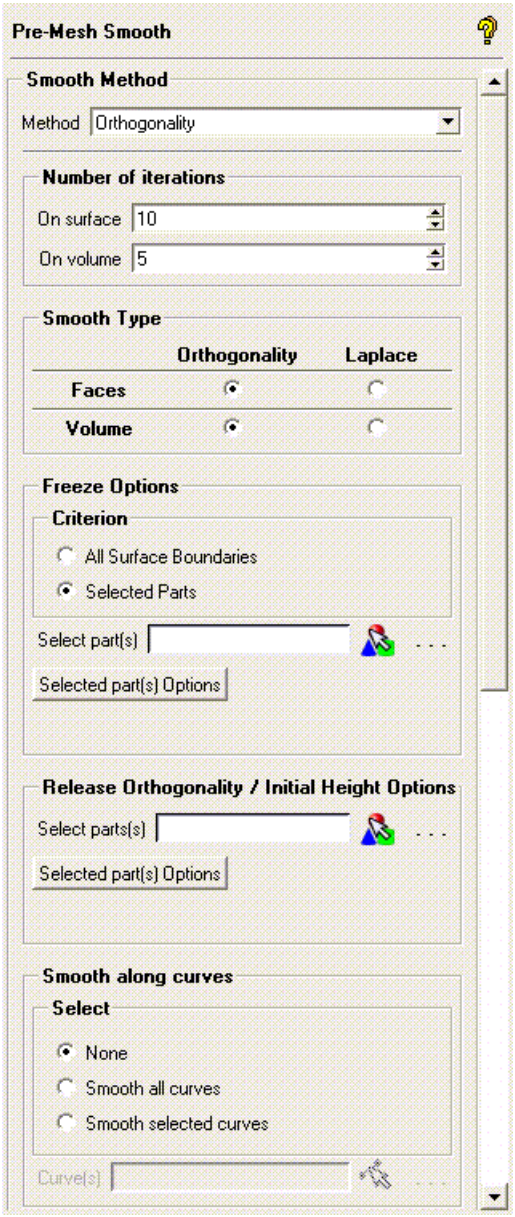
The unstructured hexahedral smoother is also available under **Edit Mesh >Smooth Hexahedral Mesh Orthogonal** to smooth an existing mesh.

Tips to improve smoothing success:

- Create the best possible starting condition by matching edge distributions.
- Create points (geometry) at the ends of the edges, and then associate the vertices (block) to them. This may be used to contain an area of fine mesh along a wall, preventing it from being totally smoothed out.

The following smoothing parameters are available for the Orthogonality method:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Number of iterations on surface

specifies the number of iterations the smoother uses to relax the surface mesh (minimum is 10). The default number of iterations on surface is also 10.

Number of iterations on volume

specifies the number of iterations the smoother uses to additionally relax the volume mesh after surface smoothing has been finished (minimum is 0). When the number specified is greater than 0, a volume smoothing step will be performed after each surface smoothing step to adjust the volume mesh to the surface mesh. The default number of iterations on volume is 5.

Smooth Type

If **Orthogonality** is selected, then the smoother tries to retain orthogonality and the height of the first layer (a different height can be specified for different parts using **Release Orthogonality / Initial Height Options**). If **Laplace** is selected, the smoother tries to equalize the mesh by setting the control function f to 0 in the elliptical differential equation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freeze Options

freezes the specified areas of the surface boundary. You can freeze all surface boundaries, or selected parts. To freeze **Selected Parts**, select the part(s), then click **Selected part(s) Options**. Specify the frozen surface boundary by enabling **Surface boundary**. Specify the number of layers to be frozen with the surface boundary by disabling **Surface boundary** and entering the number of layers n in the **Layers** field. The number of layers is set to 0 by default. When **Layers** is greater than 0, then the surface boundaries including the first n layers from the boundary will be frozen. For example, a value of 3 indicates that three layers of nodes away from the surface will be frozen along with the surface boundary.

Release Orthogonality / Initial Height Options

For certain situations, it may be helpful to release the orthogonality requirement from a certain surface part or set the first layer distance (initial height of the first element off the wall) on certain surfaces. These are mutually exclusive since orthogonality is required to set the first layer height. Select the desired parts, and then click **Selected part(s) Options**. The default is to release orthogonality for each part. You can set the initial height by disabling **Release orthogonality** and then set the initial height by entering it in the **Initial Height** field.

Note

You can set the **Initial Height** to **-1** (default) in order to keep the starting initial height.

Smooth along curves

allows you to specify which edges (projected onto curves) should be smoothed. The smoothing will be carried out when faces are smoothed. You can specify none, all, or selected curves. The default is all.

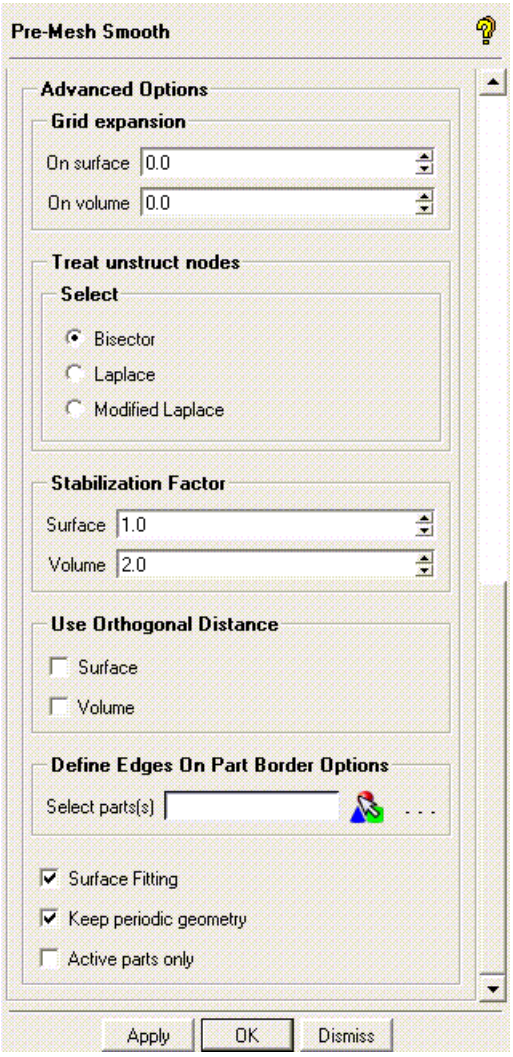
Note

If the mesh distribution has a relatively large dynamic range of mesh sizes along a curve, then selecting this option may be counter productive.

Advanced Options

contains the following advanced smoothing options:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Grid expansion rate

If a smoothing method other than Laplace is selected, then the grid expansion values will be used to distribute the control function values into the inner volume. If the grid expansion rate is greater than 0.0, the algorithm computes a pseudo structured region in which the control function values will be interpolated by an exponential decay (the grid expansion rate will then control the negative exponent (e power -g) of the control function values). If it is 0.0 then a Laplace interpolation of the control function values will be done, i.e., for an inner node, the control function value will be averaged by the control function values of its neighbor nodes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default values are 0.0 for the surface mesh and for the inner volume. Reasonable values other than 0.0 should be greater than 1.0.

Note

It may be difficult to form the pseudo structured regions in some cases. If you have such problems, set the grid expansion rate back to 0.

Note

If the grid expansion rate is set to 0.0 and the smooth type is set to **Orthogonality**, the first layer height and the orthogonality at the boundary are still achieved, but the next layers tend towards Laplace approaching the middle of the mesh. In case of a Navier- Stokes mesh you can see the effect near the boundary where the first layer is orthogonal to the boundary but the next layers curve as they equalize (Laplace) the distances of the layer nodes.

Treat unstruct nodes

determines the method of treating inner unstructured nodes. An unstructured node is a node on which no ij (surface) or ijk (volume) directions can be defined, e.g. a node which has more or less than 4 (surface) or 6 (volume) neighbors is clearly an unstructured node.

Note

For most models, the effect of the this setting may be negligible.

Stabilization Factor

is used in the calculation of the new nodes and should be greater or equal to 1.0. A higher factor will make the smoother more stable at the cost of orthogonality at the boundaries. If there are problems with the smoother it is recommended to increase this value (to around 8.0). Default values are 1.0 on surfaces and 2.0 in the volume. Increasing this parameter may be helpful on certain model configurations if the smoother corrupts the mesh, especially if it appears due to overly orthogonal boundaries.

Use Orthogonal Distance

if enabled, then the original boundary distance of a first layer node will be calculated by projecting it to the boundary and measuring the distance to the projected point. Otherwise, the length of the original grid line will be used. By default this option is disabled for both the surface and volume.

In cases where there is a sharp angle between the grid lines from the first layer to the boundary, if **Use Orthogonal Distance** has not been set, the calculated first layer distance may be considerably greater than the distance of the first layer nodes to the boundary (because the length of the grid line will be used). Hence, it may be advisable to set **Use Orthogonal Distance** in the same way for both **Surface** and **Volume**. This would ensure that the first layer grid line angles in the volume near the surface boundary would be similar to the nearby surface boundary grid line angles.

Define Edges On Part Border Options

allows you to define the border grid lines attached to the selected parts as edges, and the node distribution on the edges will be frozen. The main purpose for defining these edges is that they

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

separate the two sides of the surface mesh. If an edge is topologically internal then orthogonality will not be established there.

Surface Fitting
constrains boundary nodes to the true geometry surfaces. When disabled, the original quad faces are used to determine the boundary constraints. This option is enabled by default.

Keep periodic geometry
when enabled, allows you retain original dimensions on periodic faces. With this option disabled, point projected nodes may be moved on the corresponding surfaces during smoothing with the orthogonal smoother. This option is enabled by default.

Active parts only
if enabled, only the active parts (parts that are turned on (activated) in the **Parts** branch of the display tree, see the Parts (p. 180) section) will be smoothed. The geometry or mesh of the active parts does not need to be displayed. When this option is disabled, the whole mesh will be smoothed. This option is disabled by default.

Multiblock Method

The Multiblock method is used to smooth multiblock mesh. This smoother works on the base of the (structured) blocks of the pre-mesh.

The Multiblock smoother is used to obtain smooth grid lines. It is specially optimized for blade configurations. The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The following parameters are available for this method.

Relaxation Block(s)
allows you to select/deselect blocks. Click on the appropriate selection icon to select or deselect blocks.

Block Display Off
if enabled, the selected blocks will not be displayed on the screen.

Smoothing Direction

3D Direction
smoothes all the blocks of the pre-mesh independently.

Select direction
allows you to select the smoothing direction.

Reference Edge
specifies the edge defining the smoothing direction. To select the smoothing direction click on the Edge selection icon. A block will be highlighted in the display. Click on the edge that defines the desired smoothing direction.

If **I**, **J**, or **K** is selected, and no **Global vertices** are selected, then the blocks of the pre-mesh will be smoothed in planes starting and ending at the indices specified, and with the increment specified. The planes in between will be interpolated and smoothed, if **Volume itera-**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions is set greater than 0. If directions **I**, **J**, or **K** is selected, and the **Global Vertices** option is selected, then the pre-mesh will be smoothed globally over selected edges defined by its end vertices, in planes starting and ending at the indices specified, and with the increment specified. The planes in between will be interpolated and smoothed, if **Volume iterations** is set greater than 0.

For example, if you select a smoothing direction **I**, **Start Index** = 1, **End Index** = 20, and **Increment** = 5, the nodes on the planes I = 1, 6, 11, 16, and 20 would be smoothed with the remaining nodes interpolated. It is also possible to define a list of planes by specifying the plane numbers separated by spaces in the **Planes** field. In this case the plane numbers have higher priority than the Start and End Index fields.

Note

To identify planes, use the Blocking Display Tree > Pre-Mesh > Scan Planes function.

Auto Setup

When **Auto Setup** is selected, all subfaces which fulfill the following conditions will be selected as **Non-Relaxation Faces** and **Hold Cell Height Faces**:

- Subfaces that belong to the selected blocks.
- Subfaces that are not perpendicular to the selected smoothing direction.
- Boundary subfaces with respect to the selected blocks. For example, they cannot be block interfaces between two selected blocks.
- Subfaces that are not periodic.

Additionally, all vertices which are connected to the selected blocks will be defined as **Global Vertices**.

Layer Options

These options allow you to select layer options when smoothing.

Start Layer

is the index plane in smoothing direction at which the smoothing should start. The value should be between 1 and less than or equal to the maximum index. If plane numbers have been set under **Planes** then this field will not be used.

End Layer

is the index plane in smoothing direction at which the smoothing should end. The value should be between 1 and less than or equal to the maximum index. If plane numbers have been set under **Planes** then this field will not be used.

Increment

specifies the increment to smooth the planes between the Start and End Layers. The nodes will be interpolated in between the layers.

Volume iterations

if set to greater than, 0 then only the nodes between the planes selected for smoothing will be smoothed with this number of iterations after interpolation.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Planes
specifies the index planes in the smoothing direction for smoothing. The values should be between 1 and less than or equal to the maximum index, and should be separated by blanks. If specified then it will overwrite the values for Start and End Layer, and Increment.

Face Options
allows you to choose from the following options for selected subfaces:

Non-Relaxation Faces
freezes subfaces.

Hold Cell Height Faces
orthogonality and first cell height will be obtained on all grid lines perpendicular to the selected subfaces.

Face Display Off
if enabled, then the selected faces will not be displayed.

Face Icons
enables the display of the selected faces by icon. Otherwise the selected faces will be shown in solid display, at 90% of their full size.

Face Filters
allows you to filter the selected faces using the **Use All Selected Faces**, **Use Only Block Interfaces**, and **Use Only Boundaries** options.

Remove periodic faces
if enabled, the periodic faces will be removed from the selection.

Select or deselect faces by clicking on the appropriate selection icon.

Vertices Options

Global Vertices
If any end vertices of a block edge have been selected, and a smoothing direction is selected, then the pre-mesh will be smoothed globally in planes. It is recommended to select all vertices as **Global Vertices** and to freeze the real (not periodic) boundaries.

Layer Vertices
This option is intended to be used for O-grid vertices. For neighbor nodes, the bisector will be calculated placing the neighbor nodes on the bisector line in a distance specified as the **First Layer Distance**. If the **First Layer Distance** is smaller than 0.0, the selected vertices will not be used. Vertices can be added or removed from the selection list, and the **First Layer Distance** can be changed for the selected vertices.

Layer Vertices are only used in global edge smoothing. To access this option, a smoothing direction must have been selected.

Vertices Display Off
if enabled, the selected global or layer vertices will not be displayed.

Select or deselect vertices by clicking on the appropriate icon.

Clear all selections
resets all selections.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Options

contains the following advanced options:

Number of Iterations / Steps

Iterations in equation solver

specifies the number of iterations for the equation solver (default is 25). An equation solver (SOR method) is used to solve the elliptic equation method.

Steps in global edge smoothing

specifies the number of smoothing steps in the global-edge smoothing method. In each step the Pre-Mesh will be smoothed globally and the output will be used as input in the next step (default is 25).

Parameters

Projection limit

is useful for near-wall layers. This parameter is used for node distribution whose first spacing from the surface is less than the geometry tolerance. A value greater than 0.0 will define a distance normal to the surface up to which the nodes will be interpolated. A single value of projection limit is used at all locations on the model.

Relaxation factor

is a factor to stabilize the smoothing value. Reasonable values should be between 0.0 (exclusive) and 1.0 (inclusive). The default value is 0.5.

Residual factor

is used during global edge smoothing. Starting from the 2nd global edge step, the average change compared to the previous step will be calculated by dividing this value by the value calculated in the 2nd step. If this relative value gets lower than the **Residual factor**, a stop criterion has been reached. The default value is 0.05.

Surface fitting

constrains boundary nodes to the true geometry surfaces. With this option disabled, the boundary nodes will be projected to the triangulation of the geometry surfaces.

Use projection

enables smoothing steps to be performed, allowing nodes to move away from constraints (curves, surfaces), and then the nodes are finally projected back to the curve or surface. By default, this option is disabled.

Use orthogonal positioning

is used during global edge smoothing or plane smoothing. If **Hold Cell Height** has been set on a subface, a special method will be used to calculate the control function values in a way that the first layer nodes will be placed to hold the original or user defined (layer vertices) cell height and to be orthogonal (bisector) to the boundary. By default, this option is enabled.

Use fractional positioning

is used during global edge smoothing or plane smoothing. If **Use orthogonal positioning** has been set, the first layer nodes will not be moved in one step to the orthogonal position but in a certain amount of steps (10). This is mainly to stabilize the orthogonal positioning algorithm in highly clustered meshes. By default, this option is disabled.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Methods

In the structured smoother, several elliptic relaxation methods are available both for the volume and faces (subface boundaries). The default method is **Sorenson - Laplace**. In the case of global edge smoothing, this parameter is not relevant.

- **Sorenson-Laplace**

  Sorenson methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. This hybrid method attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior

- **Sorenson-Thomas & Middlecoff**

  This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

- **Thomas & Middlecoff**

  This method generally improves the orthogonality of grid lines across boundaries while holding the original clustering in the interior.

- **Laplace**

  This method attempts to give a uniform mesh size for all selected elements or to give a uniform transition.

- **Interpolation**

  This uses an algebraic transfinite interpolation method with Soni interpolants to generally improve internal angles.

- **Hilgenstock - Thomas & Middlecoff**

  Hilgenstock methods maintain orthogonality. This hybrid method maintains orthogonality between block boundaries (subfaces) to give a smooth transition across subfaces, while maintaining the first layer height from the boundary surface.

- **Hilgenstock - Laplace**

  This attempts to improve orthogonality and uniform node distribution within the mesh.

Grid expansion rate

is the exponent for the exponential decay of Sorenson terms from the boundary to the interior of a face. Reducing this factor will cluster the elements closer to the boundary. Default values are 3.5 for faces and 4.6 for the volume.

Multiblock Settings

Load Settings

allows you to load previously saved Multiblock settings.

Save Settings

allows you to save the current Multiblock settings in a file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Run in sequence
allows you to load different Multiblock settings files and run them in sequence. Click **Start sequence** to begin running the sequence of files.

Convert old settings
allows you to convert Multiblock settings which were created with the ANSYS ICEM CFD 4.3 version.

Convert file
specifies the old settings file.

Save as file
specifies the new settings file.

Convert
starts the conversion.

Block Checks

The **Block Checks** option allows you to check the blocks. The following methods are available for checking/fixing blocks:

Run Check/Fix
checks the internal data structures for inconsistencies and fixes them if possible.

Fix Inverted Block
fixes all the Inverted blocks. Inverted blocks have a negative determinant.

Invert Selected Block Method
inverts the selected blocks.

Pack Block Numbers
automatically renumbers all blocks in sequential order. This is to avoid very large block numbers that can result after repeated block editing and VORFN rebuilds.

Delete Block

The **Delete Block** option allows you to delete the blocks from the topology. The **Delete permanently** option is disabled by default.

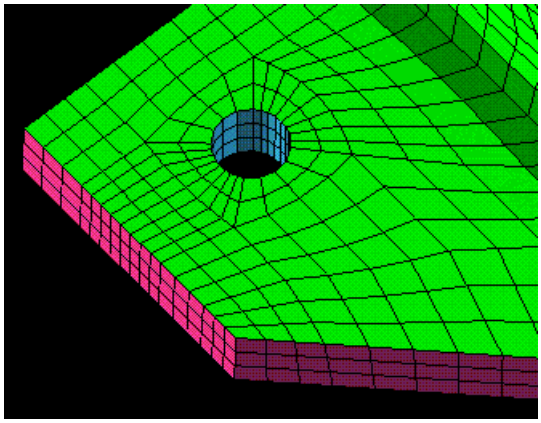
If **Delete permanently** is disabled, the block will be moved to the **VORFN** part. The VORFN part is essentially dormant and mesh is not computed for it, however it does help to maintain connectivity. Blocks can be retrieved from the VORFN region using the **Add Blocks to Part** option.

If **Delete permanently** is enabled, the block will be permanently deleted. At this point connectivity will be broken and the VORFN region will be rebuilt. This can be useful in some situations, but this will also result in a re-indexing of the blocking. The resulting index structure is usually more complicated and may be more difficult to work with.

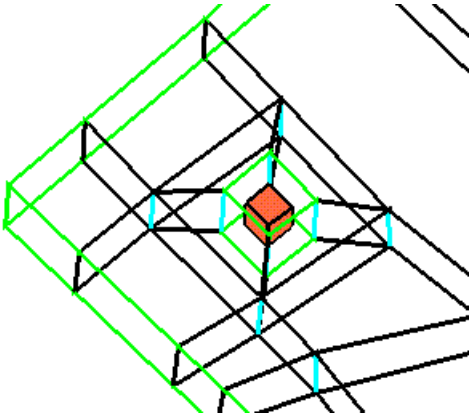
Figure 400: Example of Delete Blocks Permanently

Note that there are an equal number of nodes across the hole.

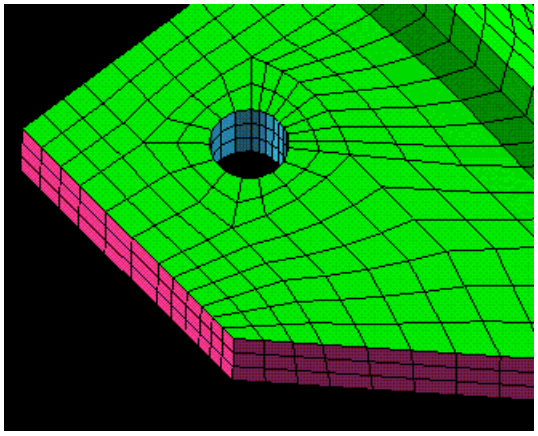
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Block selected to be deleted.



After the block is deleted permanently, the edges on opposite sides of the hole are no longer connected by a VORFN block and it is possible to adjust the edge distributions separately.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit Mesh

Figure 401: Edit Mesh Toolbar

The **Edit Mesh** menu contains all of the operations necessary to manipulate, check, improve the quality of the mesh and fix any problems.

Create Elements
Extrude Mesh
Check Mesh
Display Mesh Quality
Smooth Mesh Globally
Smooth Multiblock Domains Globally
Smooth Hexahedral Mesh Orthogonal
Repair Mesh
Merge Nodes
Split Mesh
Move Nodes
Transform Mesh
Convert Mesh Type
Adjust Mesh Density
Renumber Mesh
Assign Mesh Thickness
Reorient Mesh
Delete Nodes
Delete Elements
Edit Distributed Attribute

Note

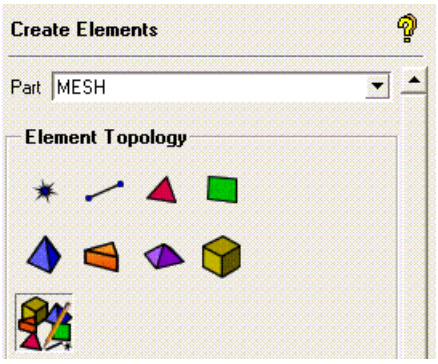
Within this menu, only the mesh elements and nodes can be checked or manipulated. You cannot manipulate or check faceted data such as STL or Nastran that represents geometry. Editing this type of data can be done in the **Geometry** menu.

Create Elements

The **Create Elements** option is used for manually creating different element types. Generally, to improve some bad elements or to get rid of some problems/errors in the mesh, some elements are deleted and good quality elements are created manually. For example, a bad quad element can be represented by two good tri elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 402: Create Elements Options



Part
Select or enter a part name. The new elements created will go into this part.

The different options for creating elements are described in the following sections.

Node (Point Element)
Bar (Line Element)
Triangle
Quad
Tetra
Prism
Pyramid
Hexa
Auto Element Type

Node (Point Element)

The **Node (Point Element)** option creates a node element. You need to select a particular position where a new node (point) should be created.

Note

This does not refer to nodes that define mesh elements, but an actual zero dimension element that is typically at an existing mesh node location. These node elements are commonly used for applying loads, constraints, and other boundary conditions.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Bar (Line Element)

The **Bar (Line Element)** option allows you to select two locations to specify a bar (line) element. It is possible to select existing nodes, points, or arbitrary locations on the screen.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

From elements
allows you to select any element(s) and line elements will be created for each edge of the element(s).

Inherit parts
if enabled, the created line elements will be added to the same part as the original elements. Otherwise, the line elements will be added to the part specified in the **Part** field.

From edges
allows you to select elements to define new line elements from their edges.

From curves
allows you to select a curve to define a new line element.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Triangle

The **Triangle** option allows you to select three locations to create a triangle. A triangular element is a surface element. If not possible, a message will be displayed that it could not create the triangle because the volume mesh could not be made consistent.

Method

From nodes
allows you to elect the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

Make Volume Consistent
automatically adjusts the volumetric elements to conform with the new element(s). This only applies when creating triangles within a tetra mesh.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Quad

The **Quad** options allows you to select four locations to specify a quad element. A quadrilateral element is a surface element. It is possible to select existing nodes, points, or arbitrary locations on the screen. Select in a clockwise or counterclockwise order unless **Automatic vertex distribution** is enabled (see below).

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

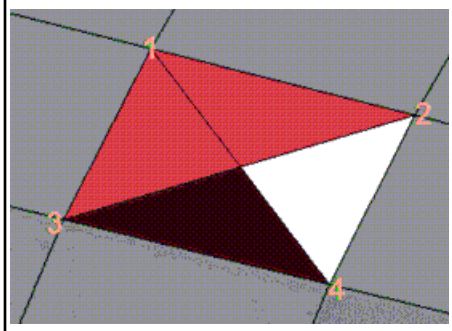
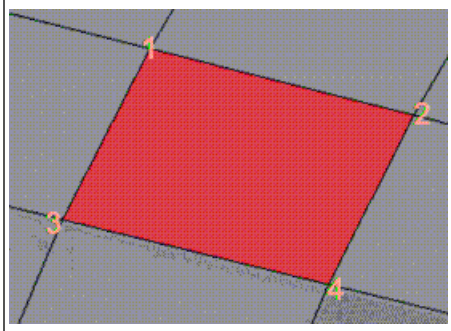
From screen
allows you to select any arbitrary location by clicking on the screen.

Make Volume Consistent
automatically adjusts the volumetric elements to conform with the new element(s). This applies when creating quads within a tetra mesh. Pyramids will be created to maintain node connectivity.

Automatic vertex distribution
For quad and hexa elements, if this option is enabled, the order of selection of nodes is disregarded, and selected nodes will be automatically reordered to avoid creating a skewed element.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 403: Example of Automatic Vertex Distribution option

Automatic Vertex Distribution disabled	Automatic Vertex Distribution enabled
	
If the nodes were selected in the order notated, a skewed element is created.	Although the nodes were selected in the same order as above, a good quality quad element is created.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Tetra

The **Tetra** option allows you to select four locations in any order to create a tetra element. A tetrahedral element has four tri faces.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Prism

The **Prism** option allows you to select six total locations to create a prism element. A prism element has two tri faces and three quad faces. Select the three locations of one face, then the three locations for the other face in the same order.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Pyramid

The **Pyramid** option allows you to select five locations to create a pyramid element. A pyramid element has four tri faces and one quad face. Select four locations of the base quad face first in a clockwise or counterclockwise order, then one location for the apex.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Hexa

The **Hexa** option allows you to select eight locations to create a hexa element. A hexahedral element has six quad faces. Select four vertices in a clockwise or counterclockwise order for one face, then four more in the same order for the opposing face. You can also select the two opposing faces.

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh file must be loaded so that nodes can be selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

From Faces
allows you to select the two opposing faces to define the hexa element.

Automatic vertex distribution
For quad and hexa elements, if this option is enabled, the selected nodes will be automatically distributed to create the best element.

Automatic element creation
automatically creates the element after selecting the point, node, or edge, without requiring confirmation by pressing the middle mouse button.

Auto Element Type

The **Auto Element Type** option allows you to create elements based on the number of vertices selected. Select any number of 1-8 vertices. Press the middle mouse button to create an element type.

Number of vertices selected	Element created
1	Node element
2	Bar (line) element
3	Tri element
4	Tetra element (if volume mesh is loaded), or Quad element (if pure surface mesh is loaded)
5	Pyramid element
6	Prism element
7	(not supported)
8	Hexa element

Method

From nodes
allows you to select the nodes of the mesh to define new elements. A mesh must be currently loaded so that nodes can be selected.

From points
allows you to select the points of the geometry to define new elements. A geometry file must be loaded so that points can be selected.

From screen
allows you to select any arbitrary location by clicking on the screen.

Make Volume Consistent
Automatically adjusts the volumetric elements to conform with the new element(s).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic vertex distribution
For quad and hexa elements, if this option is enabled, the order of selection of nodes is disregarded, and selected nodes will be automatically reordered to avoid creating a skewed element.

Extrude Mesh

The **Extrude Mesh** option allows you to create elements of one type from selected elements of a lower dimension along a defined direction. This option is generally used to create a volume mesh of non-varying cross-section from a 2D mesh profile.

Type of original elements	Type of extruded elements
Node	Bar
Bar	Quad
Tri / Quad	Prism / Hexa

The following options are common to each available method:

Elements
allows you to select the elements to be extruded.

New volume part name
specifies the part to which the new elements created during extrusion will belong.

New side part name
specifies the part to which shell elements created at the sides of the extrusion will belong.

New top part name
specifies the part to which shell elements created at the top of the extrusion will belong.

Method

The **Extrude Mesh** > **Method** is chosen from the drop down list. The options are explained in subsequent sections.

Delete options
If **Delete original elements** is enabled, the original elements selected for the extrusion will be deleted from the mesh.

If the additional option **No uncovered faces after delete** is enabled, then the original elements will only be deleted if no new uncovered faces will be created.

Extrude by Element Normal

This method of extrusion will extrude the mesh along the normal of existing mesh elements. If the normals of all the elements are not aligned in one direction, then you need to make all element normals consistent before extrusion (Edit Mesh > Reorient Mesh > Reorient Consistent).

Number of Layers
specifies the number of layers to be extruded. By default it extrudes one layer.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverse direction
when enabled, elements will be extruded in the reverse direction of the normals of existing mesh elements.

Note

The direction of the element normals will not be changed.

Spacing type
determines the height of each layer. A **Fixed** spacing will make each layer that same height. Spacing can also be defined by a **Function** of the layer variable, using Tcl syntax. See www.scriptics.com for descriptions of Tcl syntax for different mathematical expressions.

For example, Spacing can be defined as 0.3*layer. Then the mesh will be extruded along the defined vector, with the first layer height as 0.3, the second layer height as 0.6, etc.

Spacing can also be defined by a function of geometric growth as follows:

height1 * pow (ratio1, layer-1),

where height1 is the height of the first layer, pow is the expression for an exponent, and ratio1 is the growth factor.

Extrude Along Curve

This method allows you to o extrude mesh along a curve which defines variable direction. By default, the orientation of each layer will be perpendicular to the curve.

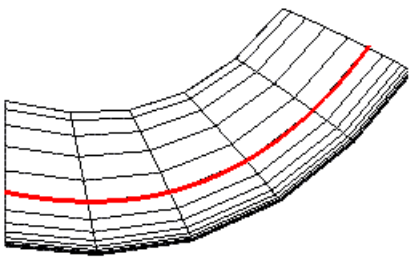
Extrude curve
specifies the existing curve along which elements should be extruded.

Show curve directions
if enabled, an arrow indicating the direction of the selected curve will be displayed.

Orient axially
The orientation of each extruded layer of elements will be the same as that of the original layer of elements.

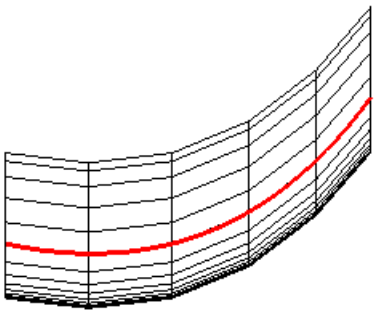
Figure 404: Extrusion Orientation

By default, the orientation of the extruded layers will be perpendicular to the curve.



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If **Orient Axially** is enabled, the orientation of the extruded layers will be the same as the original layer of elements.



Reverse curve direction
allows you to extrude elements in the opposite direction to the curve direction (parametric).

Project to Geometry
if enabled, then all new boundary nodes will be projected to surfaces. The last layer of new nodes will have the same projection type as the original nodes.

Number of layers
specifies the number of layers to be extruded. By default it extrudes one layer.

Twist per layer
specifies the number of degrees for the elements to be rotated per layer. This is used for helical extrusions.

Spacing type
You can choose a **Fixed** spacing for each element or **Curve bunching**, which will use the Spacing and Bunching law parameters that are specified for the Curve Mesh Size.

Extrude by Vector

This method allows you to extrude elements in a linear direction defined by a vector.

Direction
Choose the **Method** using the radio buttons.

Explicit Vector
specifies the X Y Z vector components of the vector, based on the active coordinate system. For example, "0 0 1" will extrude in the Z direction.

Vector by points
Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Number of Layers
specifies the number of layers to be extruded. By default it extrudes one layer.

Spacing type
determines the height of each layer. A **Fixed** spacing will make each layer that same height. Spacing can also be defined by a **Function** of the layer variable, using Tcl syntax. See www.scriptics.com for descriptions of Tcl syntax for different mathematical expressions.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, Spacing can be defined as 0.3*layer. Then the mesh will be extruded along the defined vector, with the first layer height as 0.3, the second layer height as 0.6, etc.

Spacing can also be defined by a function of geometric growth as follows:

height1 * pow (ratio1, layer-1),

where height1 is the height of the first layer, pow is the expression for an exponent, and ratio1 is the growth factor.

Extrude by Rotation

This method allows you to extrude elements by rotating them about an axis or vector.

Axis
specifies the axis or a vector to indicate the direction of rotation, or the axis to which the normal of the element is aligned.

Center of Rotation
allows you to select the origin or another point as the center of rotation.

Translate
If enabled, the extruded mesh will include both translational and rotational extrusion.

Spacing
The fixed height of each extruded layer.

Direction
Choose the **Method** using the radio buttons.

Explicit Vector
specifies the X Y Z vector components of the vector, based on the active coordinate system. For example, "0 0 1" will extrude in the Z direction.

Vector by points
Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Angle per layer
Angle of each layer of rotation.

Number of Layers
specifies the number of layers of the existing mesh for extrusion.

Merge Degenerated Elements
if enabled will merge all nodes on or near the axis of rotation within the defined **Degeneration tolerance** (the default is 0.00001). For example, degenerate hexas with one degenerate face on the axis (two sets of coincident nodes) will become prisms with one edge on the axis (two nodes).

Degeneration tolerance
If **Merge Degenerated Elements** is enabled, then this tolerance can be specified (default is 0.00001). This tolerance is used to find equal nodes in degenerated elements by using it as a ratio between the current node distance and the maximum distance of all nodes in the element.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Check Mesh

The **Check Mesh** option allows you to locate problems with the mesh that will usually lead to failure when translating or running the solution. **Errors** will most likely result in failure to write out the mesh or read it into the solver. **Possible Problems** may lead to solution crashing or diversion. You can select any combination of **Errors** and **Possible Problems** to check at one time.

The following options are available:

Set Defaults
enables the default mesh check options.

Elements to Check
You can select **All** to check the full mesh or select **Active** to check mesh from the active parts only.

Check Mode
The **Create Subsets** option will put all problematic elements in respective subsets, visually isolating them for manual editing. The **Check/fix each** option will attempt to automatically fix those problems that can be repaired. After each diagnostic check, you will be asked to respond to each problem found. If the model is very large, each check could take some time.

Interrupt
On the status bar that appears when **Check Mesh** is running, you can choose to interrupt the operation. The check that is currently running will be finished and the rest will be skipped.

Errors

Duplicate element
locates elements that share all of their nodes with another element of the same type. Duplicate elements should not exist in an ideal mesh.

Note

The deleting of elements during the automatic fix will remove one of the two duplicate elements, thus eliminating this error without creating a hole in the geometry.

Uncovered faces
For a 3D mesh, all faces on a volumetric element should either be attached to the face of another volumetric element or to a surface element. This function finds the volume elements that violate this restriction. Usually this indicates a hole in the surface mesh. In the case of a 2D mesh, this function finds edges of mesh elements that are not connected to another element or not capped off by a bar (line) element at the perimeter (single or open edges).

The automatic fix will cover these uncovered faces with triangles (surface mesh). This may or may not be the proper solution. A better method may be to first select the flawed elements and then decide if the uncovered faces are the result of missing surface mesh or the result of a hole. If it is due to missing surface mesh, rerun the check and choose the **Fix** option. If the error points out a hole in the model, you can attempt to correct the mesh by manually creating elements or merging nodes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Missing internal faces
Between every two adjacent volume elements that are in different parts there should be a surface element. This function finds the volume elements that violate this restriction. A mesh with only one volume part will not have missing internal faces. For 2D meshes, this function finds two adjacent surface elements in different parts with no line element between them. The automatic fix will create surface or line elements between these elements.

Periodic problems
checks surface parts that have periodic faces for inconsistency in the periodicity of the nodes. Errors are reported if periodic matches are missing. Slight offsets in node positions are often repaired automatically during the check process. Remaining errors can be repaired manually using Edit Mesh > Repair Mesh > Make/Remove Periodic.

Volume orientations
finds elements where the order of the nodes does not define a right-handed element.

Note

This will be automatically corrected when **Check/Fix** is run.

Surface orientations
checks for volume elements that share the same surface element. In doing this check, it is required to also check for:

- uncovered faces
- missing internal faces
- duplicate elements

This check will not find elements that occupy the same volume but are attached to regions of mesh that are not connected.

Hanging elements
checks for line elements that have one node connected to surface mesh, and the other node free (unconnected).

Penetrating elements
checks for surface elements that intersect or pass through other surface elements.

Disconnected bar elements
checks for bar elements with both nodes unconnected.

Possible Problems

Multiple edges
refers to elements with at least one edge shared among three or more elements. Legitimate multiple edges would be found at a "T" junction, where more than two geometry surfaces meet.

Triangle boxes
refers to groups of four triangles that form a tetrahedron with no volume element inside. This is best fixed by merging two of the nodes that would collapse the unwanted triangle box.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 single edges
refers to elements with two single edges. These are either corners of baffles or are triangles that are protruding from a surface and are thus undesirable in the mesh.

Single-multiple edges
refers to elements that have both single and multiple edges. These elements are probably not wanted.

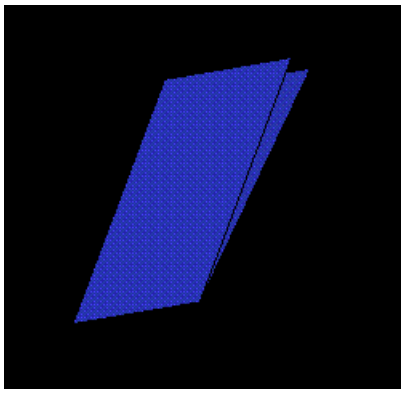
Stand-alone surface mesh
refers to surface elements that do not share a face with a volumetric element. This could be an area with an extra surface element to be deleted or a missing volume element to be created.

Single edges
refers to surface elements with at least one edge that is not shared with any other surface element. This would represent a hanging edge, and the element would be an internal baffle. These may or may not be legitimate. Legitimate single edges would exist where the geometry has a zero thickness baffle with a free or hanging edge.

Delaunay violation
refers to tri elements with nodes that are within the circumsphere of adjacent elements. These can often be removed by swapping edges of these triangles.

Overlapping elements
refers to surface elements that occupy part of the same surface area, but do not have the same nodes. This could be surface mesh that folds on to itself. This will also find elements that are at an angle of up to 5 degrees from overlapping each other, as shown in Figure 405: Example of Overlapping Element (p. 464).

Figure 405: Example of Overlapping Element



Non-manifold vertices
refers to vertices whose adjacent elements outer edges do not form a closed loop. Usually indicates elements that jump from one surface to another, forming a "tent like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the free domain that probably should not be there. See the figure in Edit Mesh > Split Mesh.

Unconnected vertices
refers to vertices that are not connected to any element. These are usually eliminated automatically upon saving the mesh.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Mesh Quality

The **Display Mesh Quality** option runs a diagnostic check of individual element quality. Mesh Quality can be displayed by a histogram. refer to the Quality Histogram section.

Mesh types to check
allows you to select the mesh types to check with a selected Quality criterion. For large 3D grids, you may want to disable the check for surface elements.

Elements to check
allows you to select the mesh elements to be checked.

- **All**

  checks all elements.

- **Active parts**

  checks all elements in active parts.

- **Visible subsets**

  checks elements in visible subsets only.

- **Visible subsets and active parts**

  checks elements in visible subsets and active parts.

Quality type
specifies the Quality criterion selected from the drop-down menu.

Refresh Histogram
refreshes the histogram displayed.

The mesh quality criteria available are described in the following sections:
Quality
Aspect Ratio
Aspect Ratio (Fluent)
Custom Quality
Determinant
Distortion
Element Stretch
Equiangle Skewness
Ford
Hex. Face Aspect Ratio
Hex. Face Distortion
Max Angle
Min Angle
Max Dihedral Angle
Max Length
Max Ortho
Min Ortho
Max Orthogls

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Ratio
Max Sector Volume
Min Sector Volume
Max Side
Min Side
Min Side (Quad Optimized)
Max Warp
Max Warpgls
Mesh Distribution
Mesh Expansion Factor
Mid Node
Mid Node Angle
Opp Face Area Ratio
Opp Face Parallelism
Orientation
Orthogonal Quality
Prism Thickness
Quadratic Dev
Skew
TGrid Skew
Surface Area
Surface Dev
Taper
Tetra Special
Volume
Volume Change
Volume/Area/Length
Workbench Shape
X Size
Y Size
Z Size
Quality Metric Histogram

Quality

The criterion **Quality** is calculated differently for different element types:

- **Tri**

  The quality is calculated as the aspect ratio of the tri element as described in Aspect Ratio (p. 467).

- **Tetra**

  The quality is calculated as the aspect ratio of the tetra element as described in Aspect Ratio (p. 467).

- **Quad**

  The quality is calculated as the Determinant, as described in Determinant (p. 470).

- **Hexa**

  The quality is a weighted diagnostic between Determinant (between -1 and 1), Max Orthogls (normalized between -1 and 1; if deviation from orthogonality is greater than 90 degrees, then

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the normalized value will be smaller than 0) and Max Warpgls (normalized between 0 and 1; warpage of 0 degrees is 1, warpage of 180 degrees is 0). The minimum of the 3 normalized diagnostics will be used.

- **Pyramid**

  The quality is calculated as the determinant.

- **Prism**

  The quality is calculated as the minimum of the Determinant and Warpage. Warpage is normalized to a factor between 0 to 1, where 90 degrees is 0, and 0 degrees is 1.

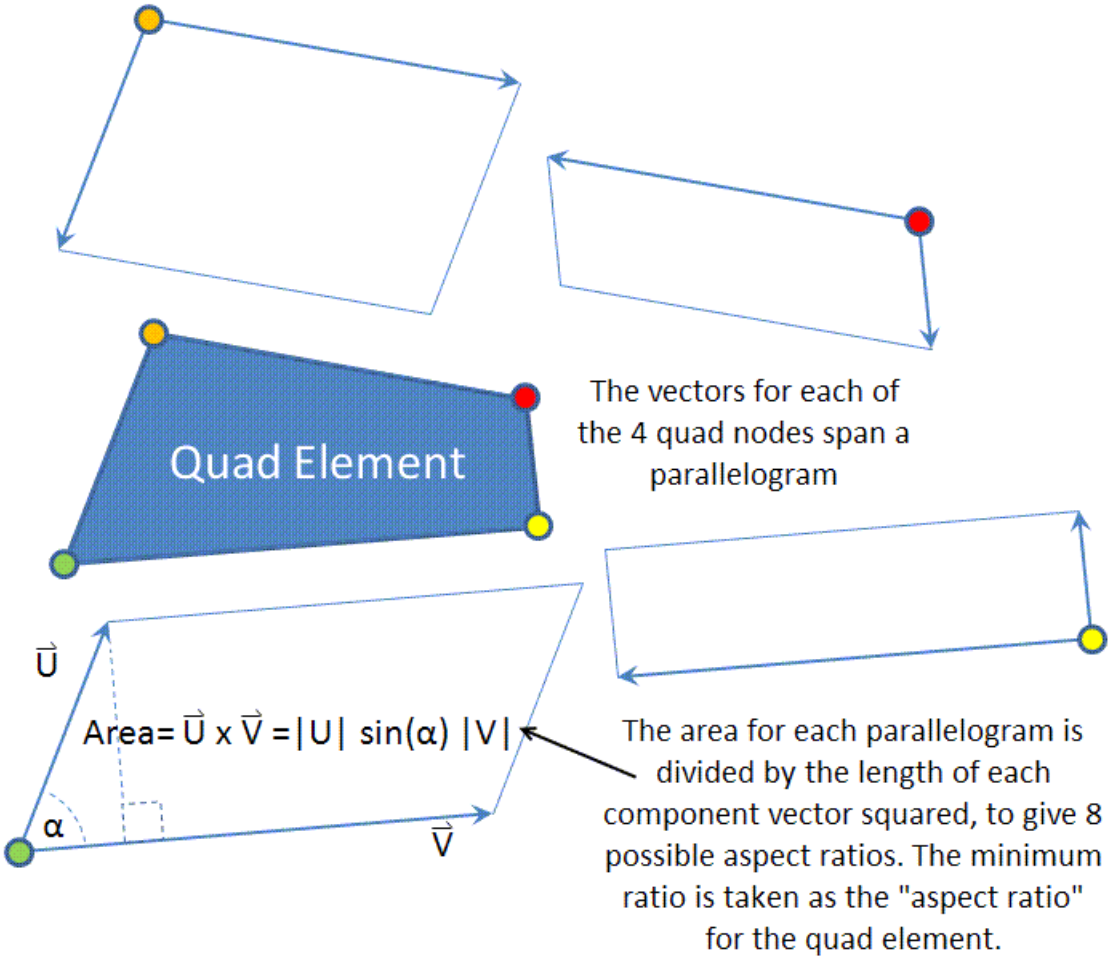
Aspect Ratio

The criterion **Aspect Ratio** is calculated differently for different element types:

- **Quad**

  The vectors for each of the 4 quad nodes span a parallelogram. The area of each parallelogram is divided by the length of each component vector squared, to give 8 possible aspect ratios. The minimum ratio is taken as the aspect ratio for the quad element.

Figure 406: Aspect Ratio for Quad Elements


Quad Element
The vectors for each of the 4 quad nodes span a parallelogram
$\vec{U}$
Area= $\vec{U}$ x $\vec{V}$ = |U| sin(α) |V|
α
$\vec{V}$
The area for each parallelogram is divided by the length of each component vector squared, to give 8 possible aspect ratios. The minimum ratio is taken as the "aspect ratio" for the quad element.

- **Hexa**

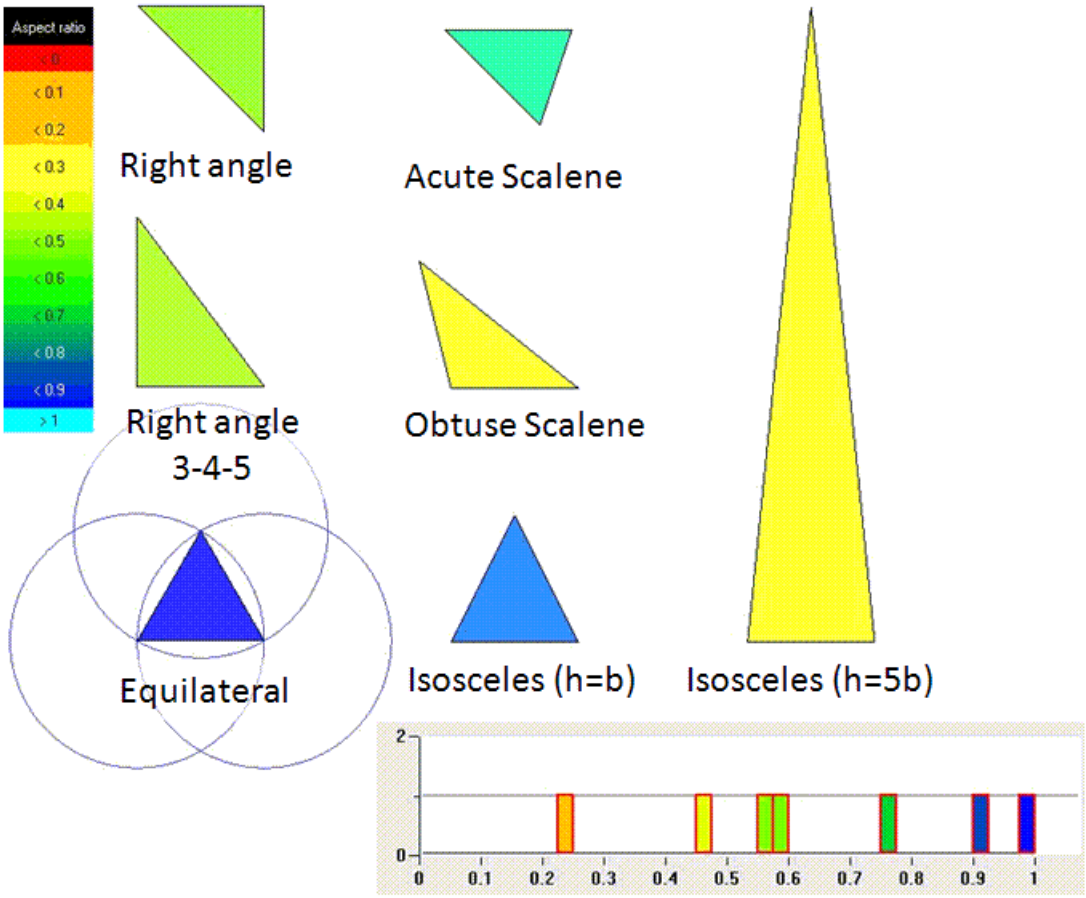
  The aspect ratio is defined by the size of the minimum element edge divided by the size of the maximum element edge. Quadratic hexahedras will also be considered.

- **Tri**

  ANSYS ICEM CFD calculates the ratio between the area of triangle and the maximum edge length for each element. The values are scaled, so that an aspect ratio of 1 corresponds to a perfectly regular element, while an aspect ratio of 0 indicates that the element has zero area.

  Aspect Ratio = (area/max edge length)$_{actual}$ / (area/max edge length)$_{ideal}$

  Figure 407: Aspect Ratio of Tri Elements–Examples



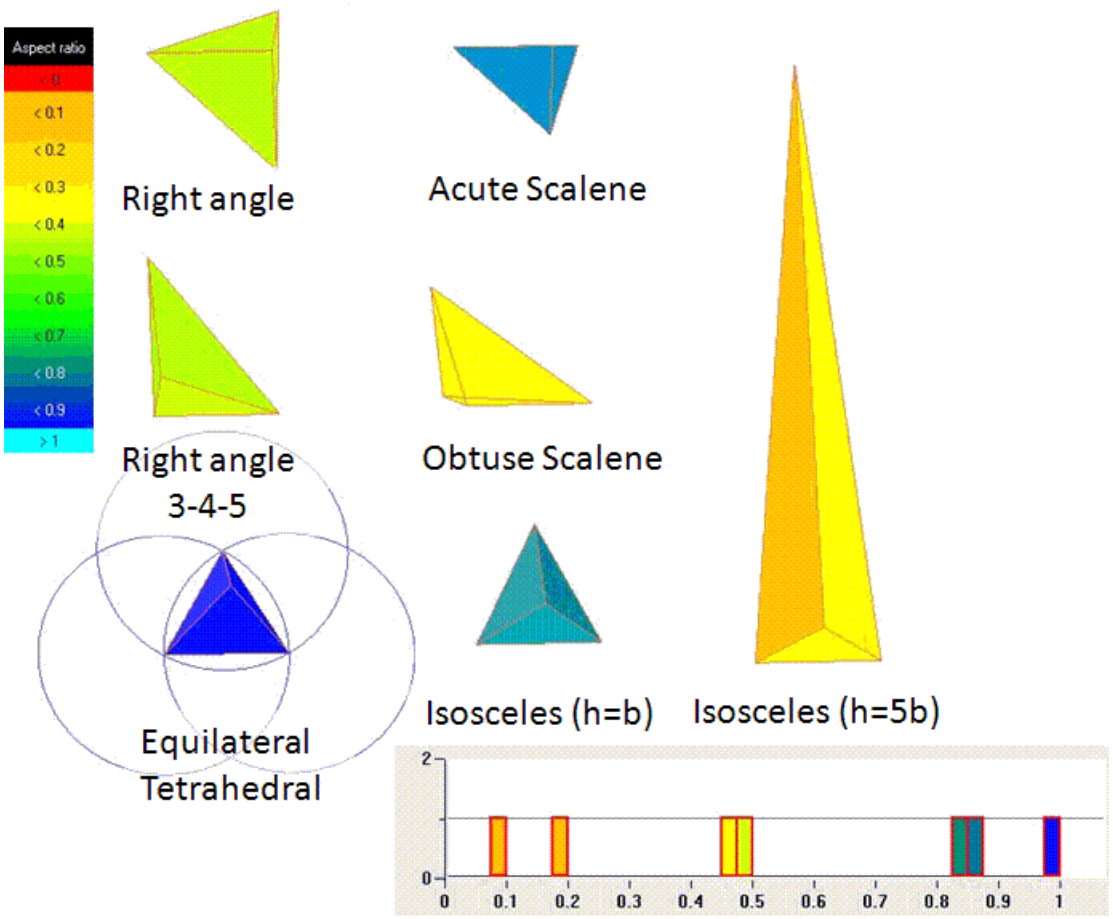
- **Tetra**

  ANSYS ICEM CFD calculates the ratio between the volume of the element and the radius of its circumscribed sphere power three. The values are scaled, so that an aspect ratio of 1 corresponds to a perfectly regular element, while an aspect ratio of 0 indicates that the element has zero volume.

  Aspect Ratio = (volume/(circumscribed radius)^3)$_{actual}$ / (volume/(circumscribed radius)^3)$_{ideal}$

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 408: Aspect Ratio of Tetra Elements–Examples

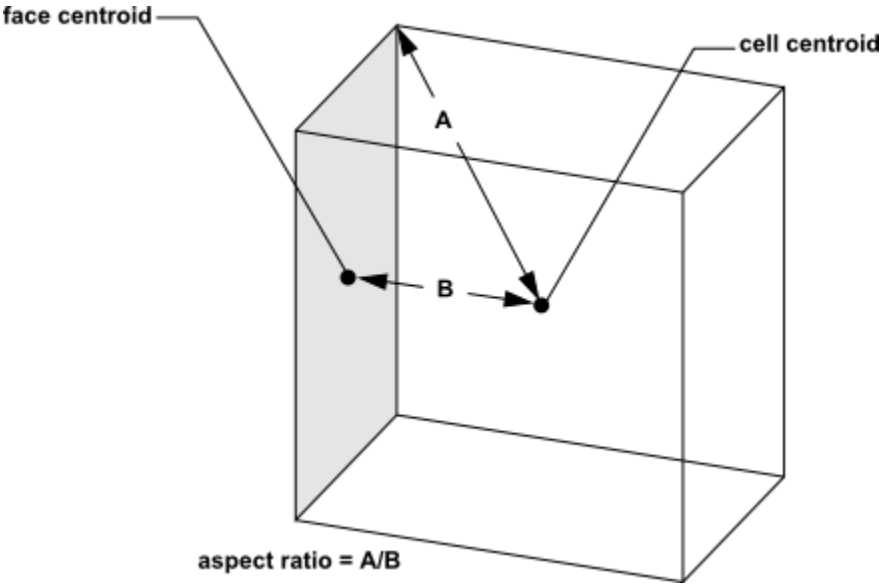


Aspect Ratio (Fluent)

This is an alternative **aspect ratio** computation (as used in ANSYS Fluent). In this case, it is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids computed as a dot product of the distance vector and the face normal, and the distances between the cell centroid and nodes. For a unit cube (see Figure 409: Calculating the Aspect Ratio for a Unit Cube (p. 470)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 409: Calculating the Aspect Ratio for a Unit Cube



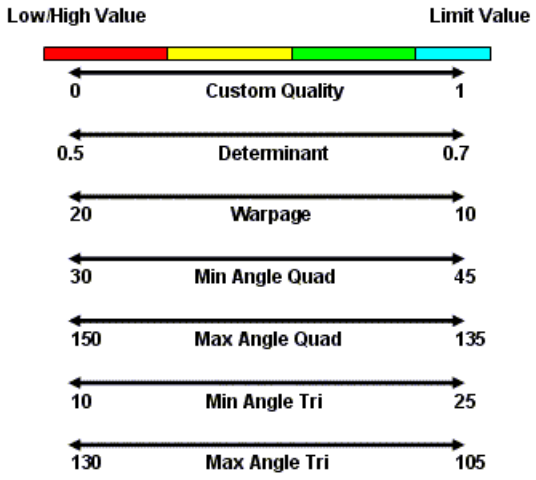
Custom Quality

This option allows you to define your own quality metric as the combination of the following quality criteria and element types: Determinant, Warp, Min Angle, and Max Angle.

You can specify the maximum and minimum quality levels of a histogram for quad and tri elements. All values are adjusted to a scale of 0 to 1 when **Recompute** is pressed.

For example, if the Min Angle Quad is 30 degrees and the maximum is 45 degrees, then 30 degrees will correspond to the minimum value in the custom quality and 45 degrees will correspond to the minimum value.

Figure 410: Custom Quality



Determinant

The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix, where each

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

determinant is computed at each node of the element. The Determinant can be found for all linear hexahedral, quadrahedral, and pyramidal elements. A Determinant value of 1 would indicate a perfectly regular mesh element, 0 would indicate an element degenerate in one or more edges, and negative values would indicate inverted elements.

Distortion

This is available only for hexa elements and is the measurement of the twisting of the element from the ideal shape. For this the Jacobian determinants will be calculated at r,s,t=-1,0,1 of the natural co-ordinate system of the element (27 node positions). The distortion is 27 times the minimum of all absolute determinants divided by the sum of all 27 absolute determinants (0 if all Jacobian determinants are 0).

Element Stretch

This option is available for surface and volume elements.

- For tris, this is calculated as the radius of the inscribed circle divided by the maximum edge length and normalized by the value of an equilateral triangle with edge length 1.
- For quads and hexas, the calculated value is the minimum edge length divided by maximum edge length normalized by the value for the idealized quad (cube) of length 1.
- For pentas and pyramids, this is calculated as the minimum of the element stretches of the quad and triangle sides.
- For tetras, this is calculated as the radius of the inscribed sphere divided by the maximum edge length normalized by the value of an equilateral tetra with edge length 1.

Equiangle Skewness

This quality parameter applies to tetra, hexa, quad, and tri elements.

Element equiangle skew = $1.0 - \max((Q_{max} - Q_e) / (180 - Q_e), (Q_e - Q_{min}) / Q_e)$,

where

Q_{max} = largest angle in the face or element

Q_{min} = smallest angle in the face or element

Q_e = angle of an equiangular face or element (e.g., 60 degrees for a triangle, and 90 degrees for a square).

Ford

A hybrid quality parameter for 3 and 4 node element meshes based on weighted skewness warp, and aspect ratio values. The possible range is from 0 to 32.

Hex. Face Aspect Ratio

This quality parameter calculates the 3 averaged face areas of hexahedra elements, which is the average of the areas of two opposite faces. The maximum of the six possible divisions of the averaged face areas will be calculated and then inverted to normalize the result.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hex. Face Distortion

This quality parameter calculates the product of the maximum edge size in the i direction multiplied by maximum edge size in the j direction, multiplied by the same in the k direction, and then divides this value by the volume of the hex element.

Max Angle

This calculates the maximum internal angle of the quad or tri faces of elements.

Min Angle

This calculates the minimum internal angle of the quad or tri faces of elements.

Max Dihedral Angle

This is the maximum angular space contained between planes which intersect. It is measured by the angle made by any two lines at right angles to the two planes.

Max Length

Calculate the maximum length of the diagonals of quad faces, and the maximum side length of tri faces. This works for all meshes and element types.

Max Ortho

This calculates the maximum deviation of the internal angles of the element from 90 degrees.

Min Ortho

This calculates the minimum deviation of any interior angles of the element from 90 degrees.

Max Orthogls

This calculates the maximum deviation of the internal angles of the elements from 90 degrees. For elements other than hexas this diagnostic is equal to Max Ortho. For hexas this differs from Max Ortho in the way that angles between 180 and 360 degrees are also considered (deviation up to 270 degrees).

Max Ratio

This calculates the maximum ratio of the lengths of any two edges that are adjacent to a vertex in an element.

Max Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the maximum of the calculated sector volumes will be taken.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min Sector Volume

This option is available for volume elements. For each element node the sector volume will be calculated in the Gauss integration points of order 3 and the minimum of the calculated sector volumes will be taken.

Max Side

This calculates the maximum length of all the edges of quad or tri faces. This works for all meshes and element types

Min Side

This calculates the minimum length of all the edges of quad or tri faces. This works for all meshes and element types

Min Side (Quad Optimized)

This calculates the minimum length of all the element edges. For quad elements, the opposite edge of the minimum element edge will also be considered e.g., if the length of the opposite edge is smaller than 1.1 times the minimum element edge length, the value will be the opposite edge length, else the value will be 1.1 times the minimum element edge length.

Max Warp

This calculates the maximum warp (in degrees) of all elements. This works only for structured volume and surface meshes, linear hexahedral and linear quadrahedral elements. To determine warp of a quadrilateral face, find the midpoints of all edges, which will be co-planar. Then, calculate the maximum angle of any edge with the plane thus defined, which is the warp of the face. The maximum warp of a volume element or hexahedron is the maximum warp of its faces. For a 2D, planar mesh of **QUAD_4** elements, the warp will be zero for all elements.

Max Warpgls

This measure applies to quad, prism and hexa elements (for quadratic elements the linear part will be checked).

To determine the warp of a quadrilateral face, the angles between the triangles connected at the 2 diagonals of the quad will be calculated and the maximum will be used.

The maximum warp of a volume element (prism or hexa) is the maximum warp of its quad faces.

For a 2D planar mesh of QUAD_4 elements, the warp will be zero for all elements.

Mesh Distribution

The mesh distribution for all elements intersected by the line defined by the **Start Point** and **End Point** locations will be calculated and displayed in the **Mesh Distribution** window. **Distribution** indicates the criterion selected. You can display the distribution for element volume (**Volume**), average element edge length (**Avg Edgelen**), minimum element edge length (**Min Edgelen**), or maximum element edge length (**Max Edgelen**). The distribution is displayed as a graph where the abscissa (x-axis) indicates the distance of the element centroid (projected onto the line) from the **Start Point** and the ordinate (y-

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

axis) indicates the selected criterion (**Volume**, **Avg Edgelen**, **Min Edgelen**, or **Max Edgelen**). This diagnostic is available for linear, tetra, pyramid, penta, and hexa elements.

Figure 411: Mesh Distribution

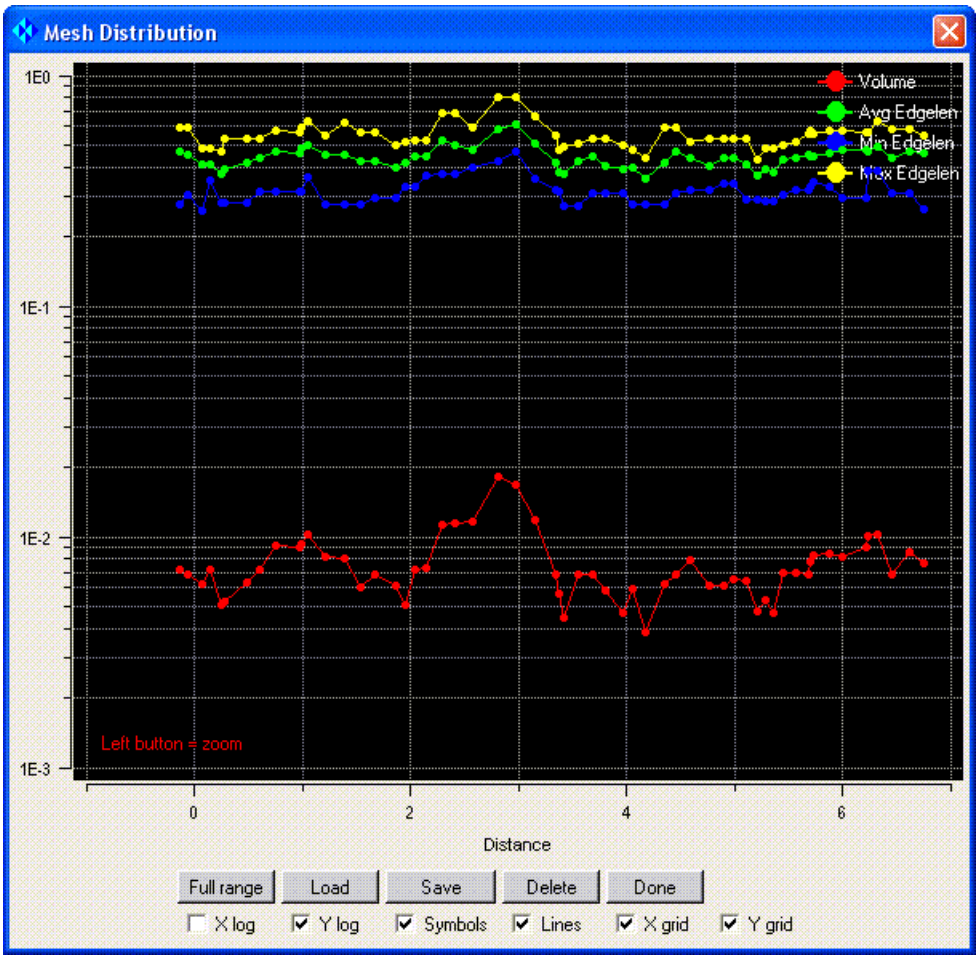


Figure 411: Mesh Distribution (p. 474) shows an example of the mesh distribution displayed. You can use the left mouse button to zoom a portion of the plotted distribution.

Full range
resets the plot in the **Mesh Distribution** window to show the full range.

Load
allows you to load the mesh distribution data from a file.

Save
allows you to save the mesh distribution data to a file.

Delete
allows you to select the plots to be removed from the **Mesh Distribution** window.

Done
closes the **Mesh Distribution** window.

X log
toggles the logarithmic scaling of the x-axis.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y log
toggles the logarithmic scaling of the y-axis.

Symbols
allows the use of symbols to mark data.

Lines
allows the use of a line to indicate the distribution data.

X grid
toggles the display of grid lines along the x-axis.

Y grid
toggles the display of grid lines along the y-axis.

Mesh Expansion Factor

This option is available for volume elements. First, the node centered sector volume is calculated for each node in the mesh. Then to calculate the expansion factor, the node centered volume is compared to the volumes around the adjacent nodes to find the largest factor. For the element quality display, for each element, the corresponding nodes will be checked and the maximum of all contributing node mesh expansion factors will be used. If there is no node contributing to the element's mesh expansion factor, then the value will be 1.

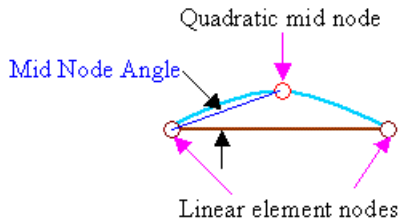
Mid Node

The mid node criterion analyzes maximum deviation of mid node.

Mid Node Angle

This criterion is applicable only for Quadratic elements. It is the angle by which the quadratic mid node is off from the linear edge.

Figure 412: Definition of Mid Node Angle

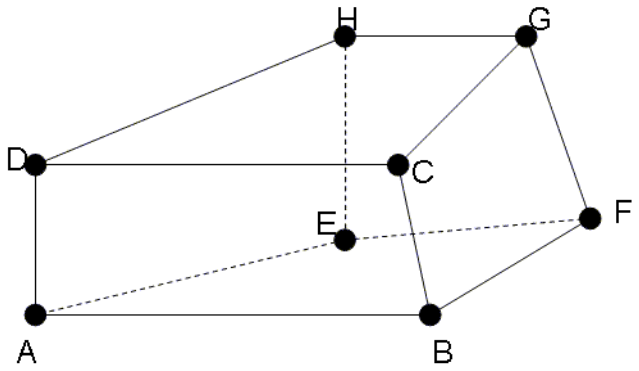


Opp Face Area Ratio

This measure is applicable to hexahedral elements only. This is the measurement of the worst ratio of the areas of the opposite faces of the hexahedral element. Ideally, this value should be 1.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 413: Definition of Opposite Face Area Ratio



In the figure, let A1 = Area of Quad Face {ABDC}, and A2 = Area of Quad Face {EFHG}. If A1>A2, then the Opposite Face Area Ratio of this pair of faces = A1/A2. If A1<A2, then the Opposite Face Area Ratio = A2/A1. Similarly, the Opposite Face Area Ratio is found for each opposing pair of faces, and the maximum of all three pairs is found as the measurement for this hexahedral element.

Opp Face Parallelism

This feature is also for hexahedral elements only. It is the measurement of the parallelism of the hexahedral elements. If the opposite faces are ideally parallel, the value is 1.

Orientation

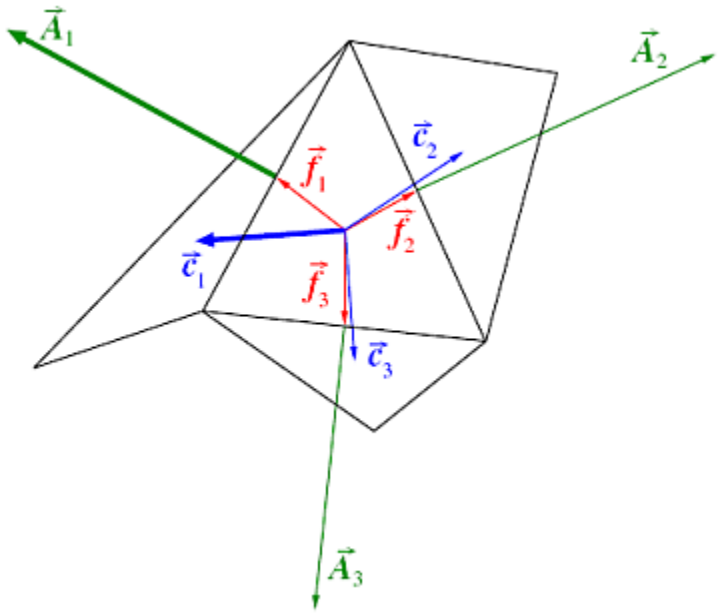
Direction of the face normal determined by the orientation of the nodes based on the right hand rule. Face orientation should be into the volumetric domain.

Orthogonal Quality

The orthogonal quality for cells is computed using the face normal vector ($\mathbf{A}_i$), the vector from the cell centroid to the centroid of each of the adjacent cells ($\mathbf{c}_i$), and the vector from the cell centroid to the centroid of each of the faces ($\mathbf{f}_i$). See Figure 414: Vectors Used to Compute Orthogonal Quality for a Cell (p. 477).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 414: Vectors Used to Compute Orthogonal Quality for a Cell



The following quantities are computed for each face:

- the cosine of the angle between the face normal vector ($\mathbf{A}_i$) and the cell centroid to adjacent cell centroid vector ($\mathbf{c}_i$).
- the cosine of the angle between the face normal vector ($\mathbf{A}_i$) and the cell centroid to face center vector ($\mathbf{f}_i$).

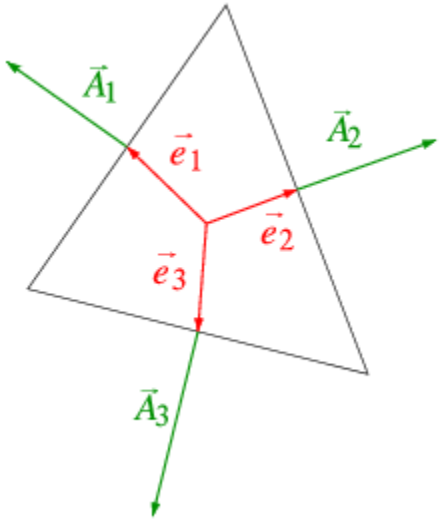
The minimum value obtained from calculating these quantities for all the faces is defined as the orthogonal quality for the cell.

The worst cells will have an orthogonal quality close to 0 while the best cells will have an orthogonal quality close to 1.

For 2D surface mesh, a similar calculation is performed using only the edge normal vector ($\mathbf{A}_i$) and the vector from the face centroid to the centroid of each edge ($\mathbf{e}_i$). See Figure 415: Vectors used to Compute Orthogonal Quality for a Face (p. 478).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 415: Vectors used to Compute Orthogonal Quality for a Face



Note

The orthogonal quality measure is related to the ortho skew measure in ANSYS Fluent except that the scale is reversed:

Ortho Skew = 1 – Orthogonal Quality

The orthogonal quality values may not correspond exactly with the ortho skew values from ANSYS Fluent due to minor tolerance differences in how the data is handled internally.

Prism Thickness

This option is available only for linear and quadratic prism elements. This calculates the average of the 3 heights at the triangle corners.

Quadratic Dev

This computes the deviation of quadratic nodes from the corresponding linear edge. Consider a triangle made up of 3 nodes: the deviation is the ratio of the altitude (the shortest line connecting the midside node to the base) with the base (the two linear nodes). A value of 0 means the midside node is on the linear edge.

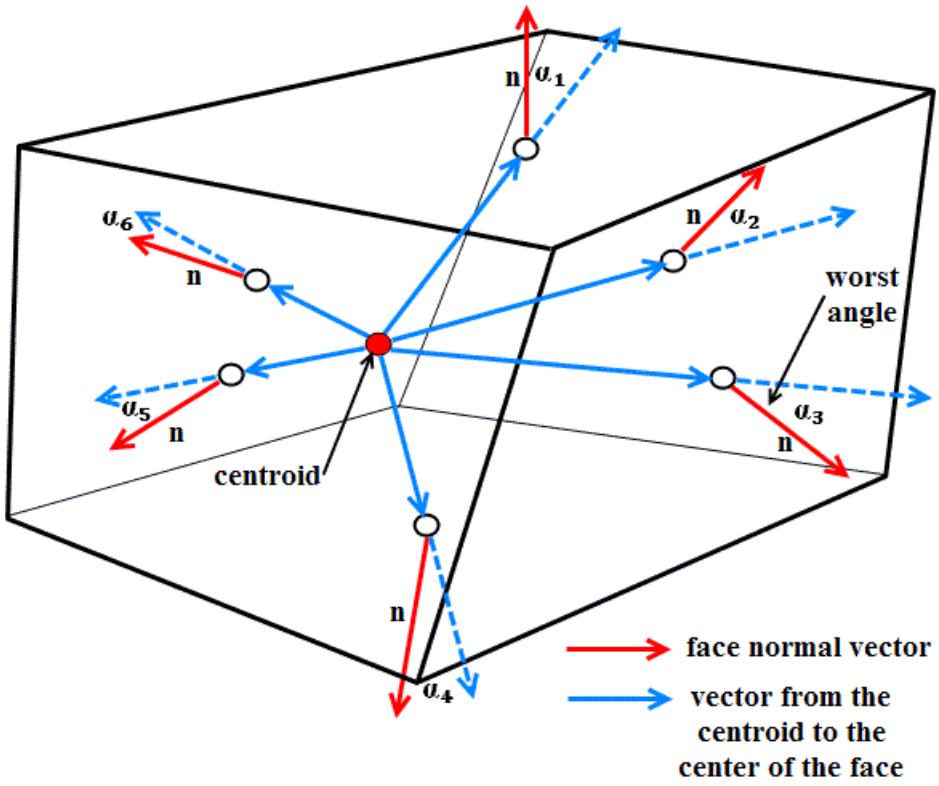
Skew

This calculates the maximum skewness of an element. The skewness is defined differently for volume and surface elements. In all cases it is normalized so that 1 is ideal and 0 is the worst possible.

- For a hexahedral element, skewness is defined as the normalized worst angle between each of the 6 face normals and the vector defined by the centroid of the hexahedron and the center of the face.

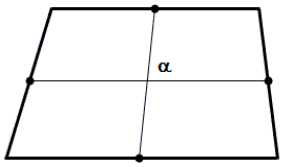
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 416: Skew for Hexahedra



- For tri elements, skewness is defined as the ratio between the area of the element and the area of an equilateral triangle having the same circumcircle.
- For quad elements, the skew is calculated by first connecting the midpoints of each side with the midpoint of the opposite side, and finding the angle α as shown in the Figure 417: Skew for Quad Elements (p. 479) (the smaller of the two angles will be used so that α<90 degrees). The result will be normalized by dividing α by 90 degrees.

Figure 417: Skew for Quad Elements



TGrid Skew

The TGrid skewness measure is computed as the normalized maximum deviation from the ideal angle at face corners (e.g., 60 degrees for a tri face, or 90 degrees for a quad face) for surface elements, or

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the normalized maximum deviation from the ideal angle between face normals (e.g., 90 degrees for a hexahedral element) for volume elements.

Note

The TGrid skewness values are inverted compared to other quality metrics in ANSYS ICEM CFD. A value of 0 indicates the best quality element while a value of 1 indicates a degenerate element.

Surface Area

This calculates the surface areas of all elements. This works for all meshes and element types.

Surface Dev

This measures the deviation of mesh from the actual geometry.

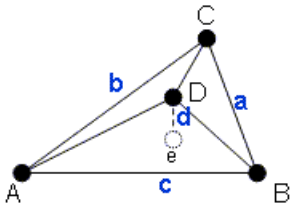
Taper

For hexahedral elements, the **Taper** is the maximum ratio of the areas of opposite faces. For quad elements, it is the maximum ratio of the lengths of opposite edges.

Tetra Special

This determines the ratio of the largest element edge to the smallest height for linear tetrahedral elements.

Figure 418: Definition of Tetra Special



In the figure, let d = height normal to surface ABC to point D. The Tetra Special value = {Max (a, b, c)}/d. The maximum value calculated from all the nodes is the Tetra Special value for the tetrahedral element.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume

This computes the volume of individual elements.

Note

This diagnostic does not consider degenerate hex elements, and will assign a negative value for them. The **Volume/Area/Length** check can be used for degenerate hex elements instead.

Note

The **Volume** and **Volume/Area/Length** criteria essentially use the same calculation for most element types and orders (linear or quadratic), but the volume check is optimized for 8–noded (linear) hex elements. It is recommended to use **Volume/Area/Length** unless you only want to check the volume mesh, or you have a linear hex mesh. In these cases, the Volume check would be faster.

Volume Change

This quality metric is calculated for a specific element by finding the maximum volume among all of its neighbor elements divided by the volume of the element itself.

Volume/Area/Length

This quality parameter calculates the line length of line elements, the area of surface elements, and the volume of volume elements, with evaluation at Gauss points (4th order). It is implemented for all supported element types, including quadratic element types.

Note

The **Volume** and **Volume/Area/Length** criteria essentially use the same calculation for most element types and orders (linear or quadratic), but the volume check is optimized for 8–noded (linear) hex elements. It is recommended to use **Volume/Area/Length** unless you only want to check the volume mesh, or you have a linear hex mesh. In these cases, the Volume check would be faster.

Workbench Shape

This option is applicable to all shell and volume elements. It provides a composite quality metric that ranges between -1 and 1. The metric is based on the ratio of the volume to edge length for volume elements and the ratio of area to edge length for shell elements. A value of 1 indicates a perfect cube or square, while a value close to 0 indicates a bad element, and a negative value indicates an inverted element.

X Size

This computes the length of the element profile in the X direction.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y Size

This computes the length of the element profile in the Y direction.

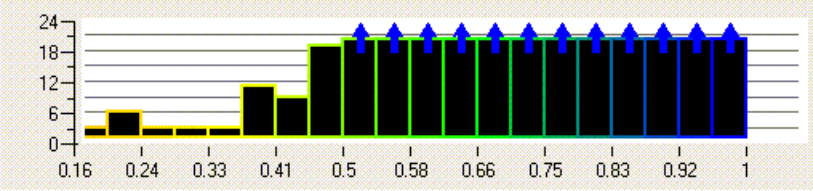
Z Size

This computes the length of the element profile in the Z direction.

Quality Metric Histogram

Mesh quality is displayed as a histogram where the abscissa (x-axis) displays the element quality on a scale from 0 (worst) to 1 (best) and the ordinate (y-axis) displays the number of elements in each quality range. The default is 20 bars or divisions between 0 and 1. So the first bar would display the number of elements whose quality is between 0 and 0.05, and the next bar between 0.05 and 0.1. The same histogram values are also tabulated in Messages window. The Status message line will display the min and max values of the quality criterion.

Figure 419: Mesh Quality Histogram

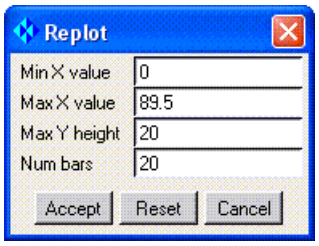


By clicking on any of the bars with the left mouse button, the information about the precise number of values that fall into this interval and its boundaries are displayed in the Message window. The color contour bar will also appear in the display. Histogram bars that have been selected become a solid color, and remain selected until clicked on again. To display the elements within a selected (highlighted) histogram bar, right click on the histogram window, and click **Show** (enabled by default). The elements represented by that bar will be highlighted. If the **Solid** option is enabled, then the color of the elements will represent the corresponding range of quality as indicated by the color contour bar.

By right clicking on the histogram, the following options are available.

Replot
allows you to change the parameters of the histogram in the **Replot** window.

Figure 420: Replot Window



- **Min / Max X value**

  sets the minimum and maximum values of the particular quality type selected for the X-axis.

- **Max Y height**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sets the maximum number of blocks for the quality ranges. A value of 0 will set the display to such that the histogram with the largest number of elements is fully visible. A smaller number will give increased resolution for quality ranges with fewer elements.

- **Num bars**

  sets the number of bars to use in the histogram display.

Reset
sets the maximum number on the Y-axis such that the histogram bar with the largest number of elements is fully visible.

Refresh
recomputes the mesh quality and refreshes the histogram display.

Show
displays the elements within the selected (highlighted) histogram bars.

Solid
displays all selected elements in solid view and by color that corresponds to the quality range. The color contour bar will display the range of quality by color.

Note

This option slows down the display speed.

Color By Quality
when enabled (default), clicking on any histogram bar will display those elements colored according to quality, as defined by the color bar which appear on the left side of the display window.

Highlight
applies to surface mesh only. Select one of the following options to highlight: selected elements only, selected elements and one layer of attached surface elements, or selected elements and two layers of attached surface elements.

Subset
puts the selected elements into a new subset.

Done
closes the histogram window.

Smooth Mesh Globally

The **Smooth Mesh Globally** option allows you to automatically improve the quality of the mesh elements. Different smoothing algorithms are available depending on which mesh type is loaded. Mesh can be smoothed with respect to a particular quality criterion and with a specified number of iterations to achieve a given quality level. A mesh containing tetras, pyramids, prisms and triangular and quad

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

surface elements can be smoothed. The element quality is displayed as described in Display Mesh Quality.

Note

It is no longer necessary to load a geometry file in order to perform mesh smoothing. Surface nodes will be constrained to the boundary represented by the original mesh. Fixed and edge constraints are also kept, as represented by node and bar elements.

If the desired quality is not attained, simply re-smooth until no further progress is seen. Then consider manual editing. Place the worst elements in a subset or select them from the histogram in order to display them and make them active for editing.

If a multiblock mesh is loaded, see **Smooth Multiblock Domains Globally** for smoothing options.

The following options are available for smoothing the mesh:

Quality

Smoothing iterations
specifies the number of iterative steps. Each iteration will apply smoothing to a percentage of the elements below the specified quality. The more iterations, the smaller the percentage of elements that is selected and incremented for each step. For example, if 5 iterations are specified, the first iteration will smooth the worse 20% of elements, the second will smooth 40%, etc. Increasing the number of iterations is more robust, but less iterations means that each iterative step is smoothing more elements each time.

Up to value
specifies the quality level up to which the smoother will attempt to smooth the mesh.

Criterion
specifies the quality criterion. The following criteria are available:

- **Quality**
- **Aspect ratio**
- **Custom quality**
- **Determinant**
- **Min angle**
- **Max orthogls**
- **Max warp**
- **Max warpgls**
- **Skew**

Refer to Quality (p. 466) for details on the quality criteria.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth Mesh Type

Smooth
If enabled for a particular element type, then this element will be smoothed in order to produce a higher grid quality. The quality of these element types will be represented in the quality histogram.

Freeze
If enabled for an element type, the nodes of this element type will be fixed during the smoothing process. This element type will not be displayed in the histogram.

Float
If enabled for an element type, the smoother may move nodes of this element type freely according to their geometric constraints (points, curves, and surfaces). Nodes of floating elements are only moved if necessary to improve the quality of the adjacent smoothed elements. In the case that the nodes are moved, the quality of the floating elements is considered with the same priority and with the same quality criterion/method as the smoothed elements. These floating elements are not included in the histogram.

Note

There is a surface smoother and a volume smoother. If you smooth the surface mesh, but float the adjacent volume mesh, you may find that the surface smoother quality checks are not sufficient to maintain good quality in the adjacent volume mesh. However, between volume mesh types or when floating surface mesh, the quality of the floating elements will be maintained.

Note

When smoothing a subset, the subset is actively smoothed while all other mesh is set to float.

Smooth Parts/Subsets

All Parts
allows you to smooth elements in all parts.

Only active parts
allows you to smooth only the parts that are activated/enabled in the Display tree.

Only visible subsets
allows you to smooth only the visible subsets.

Active parts and visible subsets
allows you to smooth both the parts enabled in the Display tree and the visible subsets.

Refresh Histogram
recalculates and updates the histogram. This is useful when changing the **Smooth Mesh Type** options and you want to simply check the quality of the elements prior to performing a smoothing step.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Options

Laplace smoothing

Pure
solves the standard Laplace equation that calculates the average of all neighbor nodes for any node and gives a more uniformly spaced mesh. This setting operates on mesh types set to **Smooth**, smoothing surface mesh types first and then volume mesh types.

Note

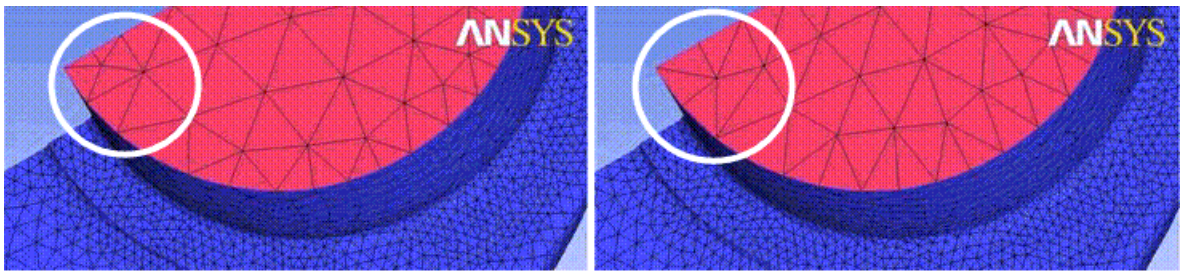
Laplace smoothing surface mesh before inflating prisms results in better prism quality. Also, using the standard Laplace equation for prisms can sometimes lead to a lower determinant quality. Therefore, it is recommended to run this prior to prism generation.

Edge length
uses a modified Laplacian scheme which tries to equalize the edge lengths around a node. This option is useful in cases where the mesh size distribution is quite wide. The smoother will try to increase the minimum edge length toward the average size to improve mesh uniformity. Figure 421: Edge Length Based Laplace Smoothing (p. 486) shows an example where the minimum edge length on the mesh increased after smoothing using the **Edge length** criterion. The highlighted portion shows a set of elements where the smoothing scheme was able to redistribute the edge lengths to be more equal.

Note

As this method tends to increase the minimum edge lengths in a model, it can be very useful for explicit solvers whose time step is tied to the minimum edge length.

Figure 421: Edge Length Based Laplace Smoothing



Residual on Surfaces
when enabled, allows you to specify the residual for surfaces.

Residual in Volume
when enabled, allows you to specify the residual for the volume.

The residual represents the smoother "convergence" rate with respect to the initial state. It is calculated as the ratio of average node movement at the current iteration to the average node movement at the first iteration. You can specify the residuals for surface and volume

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

while smoothing. The residual at each iteration will be reported in the message window. When the specified residual value is reached, the smoothing process will stop.

Note

The acceptable range for residual values specified is 0–1.

Not just worst 1%
if enabled, it will evaluate all elements while smoothing. By default the smoother only smoothes the worst 1% of each element type.

Allow node merging
if enabled, the smoother is allowed to merge nodes to improve the mesh.

Allow refinement
allows the smoother to automatically subdivide tetras to obtain further improvement. After smoothing with **Allow refinement** enabled, it may be necessary to smooth further with this option disabled. The goal of this option is to reduce the number of cells that are attached to one vertex by refinement in problem regions.

Group bad hex regions
is used only in unstructured hexa smoothing. If enabled, then bad hex elements are grouped into bad regions. The smoother will run on these bad regions one at a time. If disabled, then the smoother will smooth all the groups at once.

Ignore PrePoints
is used only for unstructured hexa smoothing. This option simply ignores the prescribed points which hinder the quality of mesh.

Surface Fitting
is used only for unstructured hexa smoothing. It forces the smoother to follow B-spline surfaces rather than the triangulation of the surfaces.

Prism Warpage Ratio
prisms are smoothed based on a balance between prism warpage and prism aspect ratio. Numbers from 0.01 to 0.50 favor improving the prism aspect ratio, and from 0.50 to 0.99 favor improving prism warpage. The farther the value is from 0.5, the greater the effect.

Violate Geometry
allows the smoothing operation to yield a higher quality mesh by violating the constraints of the geometry. Normally when a grid is smoothed, the nodes are restricted to the geometry and can only be moved along the geometric entities to obtain a better mesh.

Note

This option would be advantageous for situations such as a region where two surfaces come together at an angle that makes good element quality difficult to obtain (angles under 30 degrees). In this case, the mesh cannot both accurately capture the geometry, and give good quality mesh. The geometry (the 30 degree angle) can be sacrificed a small amount to improve the element quality if this option is utilized. A large number of bad elements can be fixed in this way by making a small sacrifice on the geometry.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tolerance
is the distance measured in units of the model that nodes can be moved off the geometry, if **Violate Geometry** is enabled.

Relative Tolerance
if enabled, this will make the specified **Tolerance** the factor of the minimum edge length of the mesh instead of an absolute tolerance in units of the model.

Minimum Edge
if enabled, you can specify the minimum edge length allowed in the mesh after smoothing.

Smooth Multiblock Domains Globally

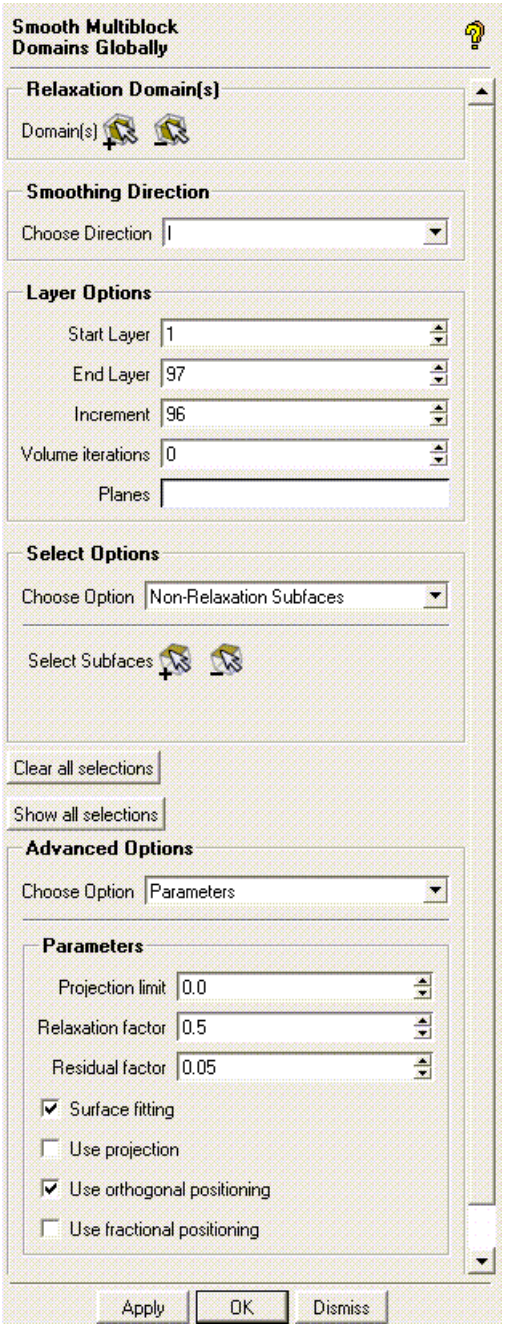
The Multiblock smoother is used to obtain smooth grid lines. It is specially optimized for blade configurations. The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The Multiblock smoother can also constrain nodes in one dimension and smooth nodes in the other dimensions in order to improve mesh quality. This is useful in turbine blade analysis e.g., for the inlet and outlet surfaces. While smoothing, the blade surfaces are frozen to maintain geometric location. Previously, the inlet and outlet would be frozen, thereby freezing the ends of the periodic sides, and thus, limiting the smoother. By constraining nodes in one dimension, the inlet and outlet are constrained axially, maintaining their distance from the blade as well as their periodicity. However, the nodes can be smoothed in the other dimensions, thereby allowing the smoother more flexibility.

For all other mesh types, the options for **Smooth Mesh Globally** will be displayed.

The following parameters are available when a multiblock mesh is loaded:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Relaxation Domain(s)

click the appropriate icon to select or deselect domain(s) to be smoothed.

Smoothing Direction

3D

smoothes the mesh in all directions.

I, J, or K

allows you to select the smoothing direction.

Layer Options

This smoother works by smoothing planes and then interpolating the smoothing between the layers. The model is reduced to a single block high and the layers are the mesh indices from one side to the

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

other along the smoothing direction. You need not smooth every plane. In models with very little twist, selecting the first and last plane is sufficient. If a model (such as a compressor blade) has more twist, it may be advantageous to smooth the first and last, as well as a few layers between.

Start Layer, End Layer
the first (usually 1) and last (usually the total number of layers) to be smoothed. These form the ends of the smoothed volume and are usually setup automatically.

Increment
allows you to add smoothing planes between the first and the last planes. For instance, if you have 26 planes, you could have an increment of 5, which would mean the 6th, 11th, 16th and 21st planes would also be smoothed. Use a smaller increment to include more planes and better capture twist or other changes in the cross section. Setting the increment to 1 will smooth each plane individually, and is usually not necessary.

Volume iterations
specifies the number of smoothing iterations between each actively smoothed layer. If this value is greater than 0 then the layers between the smoothed planes will not only be interpolated but additionally smoothed.

Planes
specifies the planes which need to be smoothed directly. If set, then the **Start Layer**, **End Layer** and **Increment** setting will not be used. The plane numbers are separated by blanks (e.g., "1 5 10")

Select Options

Non-Relaxation Subfaces
freezes the selected subfaces.

Hold Cell Height Subfaces
orthogonality and first cell height will be obtained on all grid lines perpendicular to the selected subfaces.

Fixed Edges
all edge nodes will be frozen for each selected edge.

Fixed Distribution Edges
the corresponding nodes can move but the original bunching will be retained for each selected edge.

Global Vertices
if any end vertices of a block edge have been selected, and a smoothing direction is selected, then the Pre-Mesh will be smoothed globally in planes. It is recommended to select all vertices as Global Vertices and to freeze the real (not periodic) boundaries.

Layer Vertices
is intended to be used for O-grid vertices. For neighbor nodes, the bisector will be calculated placing the neighbor nodes on the bisector line in a distance specified as the **First Layer Distance**. If the **First Layer Distance** is smaller than 0.0, the selected vertices will not be used. Vertices can be added or removed from the selection list, and the **First Layer Distance** can be changed for the selected vertices.

Layer vertices are only used in Global edge smoothing. To access this option, a smoothing direction must have been selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clear all selections
resets all selections.

Advanced Options
The following advanced options are available:

Number of Iterations/Steps

Iterations in equation solver
An equation solver (SOR method) is used to solve the elliptic equation method. This value specifies how many iterations should be done (default is 25).

Steps in global edge smoothing
specifies the number of smoothing steps in the global-edge smoothing method. In each step, the Pre-Mesh will be smoothed globally and the output will be used as input in the next step (default is 25).

Parameters

Projection limit
is used for node distribution whose first spacing from the surface is less than the geometry tolerance. It is useful for near-wall layers. A value greater than 0.0 will define a distance normal to the surface up to which the nodes will be interpolated. A single value of projection limit is used at all locations on the model.

Relaxation factor
is a factor to stabilize the smoothing value. Reasonable values should be between 0.0 (exclusive) and 1.0 (inclusive). Default value is 0.5.

Residual factor
is used during global edge smoothing. Starting from the 2nd global edge step, the average change compared to the previous step will be calculated by dividing this value by the value calculated in the 2nd step. If this relative value gets lower than the specified residual factor, a stop criterion has been reached. Default value is 0.05.

Surface fitting
constrains boundary nodes to the true geometry surfaces. With this option off, the boundary nodes will be projected to the triangulation of the geometry surfaces.

Use projection
smoothing steps are performed allowing nodes to move away from constraints (curves, surfaces), and then the nodes are finally projected back to the curve or surface. This option is disabled by default.

Use orthogonal positioning
is used during global edge smoothing or plane smoothing. If **Hold Cell Height** has been set on a subface, a special method will be used to calculate the control function values in a way that the first layer nodes will be placed to hold the original or user defined (layer vertices) cell height and to be orthogonal (bisector) to the boundary. This option is enabled by default.

Use fractional positioning
is used during global edge smoothing or plane smoothing. If **Use orthogonal positioning** has been set, the first layer nodes will not be moved in one step to the orthogonal position but in

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a certain amount of steps (10). This is mainly to stabilize the orthogonal positioning algorithm in highly clustered meshes. This option is disabled by default.

Methods

In the structured smoother, several elliptic relaxation methods are available both for the **Volume** and **Subfaces** (subface boundaries). The default method is **Sorenson - Laplace**. In the case of global edge smoothing, this parameter has no meaning.

Sorenson-Laplace

Sorenson methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. This hybrid attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior

Sorenson-Thomas & Middlecoff

This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

Thomas & Middlecoff

This method generally improves the orthogonality of grid lines across boundaries while holding the original clustering in the interior.

Laplace

This method attempts to give a uniform mesh size for all selected elements.

Interpolation

This uses an algebraic transfinite interpolation method with Soni interpolants to generally improve internal angles.

Hilgenstock - Thomas & Middlecoff

Hilgenstock methods maintains orthogonality. This hybrid maintains orthogonality between block boundaries (subfaces) to give a smooth transition across subfaces, while maintaining the first layer height from the boundary surface.

Hilgenstock - Laplace

This attempts to improve orthogonality and uniform node distribution within the mesh.

Grid expansion rate

is the exponent for the exponential decay of Sorenson terms from the boundary to the interior of a subface. Reducing this factor will cluster the elements closer to the boundary. Default values are 3.5 for **Subfaces** and 4.6 for the **Volume**.

Multiblock Settings

Load Settings

allows you to load previously saved Multiblock Settings.

Save Settings

allows you to save the current Multiblock Settings in a file.

Run in sequence

allows you to load and run different Multiblock Settings files in sequence. Click **Start sequence** to begin running the sequence of files.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

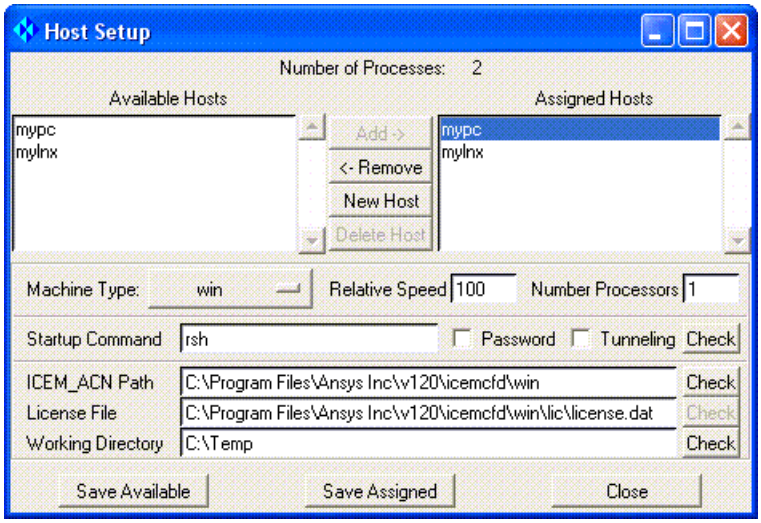
Distributed Smoothing

Distributed Smoothing

enables the use of distributed smoothing which allows you to smooth the mesh using multiple processes executing on the same computer or different computers in a network.

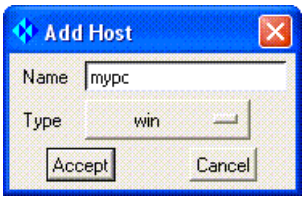
Define Hosts

opens the **Host Setup** panel where you can specify the host settings.



You need to do the following to specify the host settings:

1. Click **New Host** to open the **Add Host** panel where you can define the host machine. Specify the name of the host and select the operating system type from the **Type** drop-down list. Click **Accept**.



2. All defined hosts will be listed in the **Available Hosts** list. Select the host in the **Available Hosts** list and verify the relevant parameters.

   a. **Number Processors** specifies the number of processors on the machine.

   b. The **Startup Command** allows you to use either the remote shell client (**rsh**) or secure shell client (**ssh**). Enable **Password** if you require a password to access the machine. You may click the **Check** button to verify that the startup command works correctly.

   c. Verify that the **ICEM_ACN Path** and **License File** path are correct and specify the **Working Directory** as appropriate.

   d. Click **Save Available** to save the settings for the available hosts.

3. Select the host in the **Available Hosts** list and click **Add ->** to add the host to the **Assigned Hosts** list. To remove a host from the **Assigned Hosts** list, select the host and click **<- Re-**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

move. You can also permanently delete previously defined hosts using the **Delete Host** button.

4. Click **Save Assigned** to save the settings for the assigned hosts and close the **Host Setup** panel.

Number Processes
specifies the number of processes.

Connect Timeout
specifies a timeout for connecting to the host machine.

Verbose Print
enables the printing of detailed messages in the message window.

Smooth Hexahedral Mesh Orthogonal

The unstructured hexahedral smoother relaxes unstructured hexahedral meshes in order to obtain smooth grid lines orthogonal to the boundary as well as smooth grid angles and transitions in the inner volume. It first smoothes the surface mesh recognizing the topological boundary edges. If the number of volume smoothing steps is greater than 0, after each surface smoothing step the inner volume will be adjusted by performing 1 volume smoothing step. After the surface smoothing has been finished, the inner volume will be smoothed (according to the number of volume steps set). It can also smooth pure surface meshes.

The mathematical basis is that an elliptical differential equation of the form $\nabla^2 \mu = f$, where f is the "control function", will be solved. It can be proved that by using the elliptical operator ∇^2, smoothness of the mesh will be achieved. The control function f will be specified so that the smoothed mesh will obtain certain characteristics, such as orthogonality and layer height of the first layer.

The unstructured hexahedral smoother is available under **Edit Mesh > Smooth Hexahedral Mesh Orthogonal** (to smooth an existing mesh) and under **Blocking > Pre-Mesh Smooth** (to smooth the Hexa pre-mesh). If you select **Blocking > Pre-Mesh Smooth**, the pre-mesh will be calculated (if it does not exist), and then you can select the **Orthogonality** option for the **Smooth Method** for the unstructured hexahedral smoother.

Note

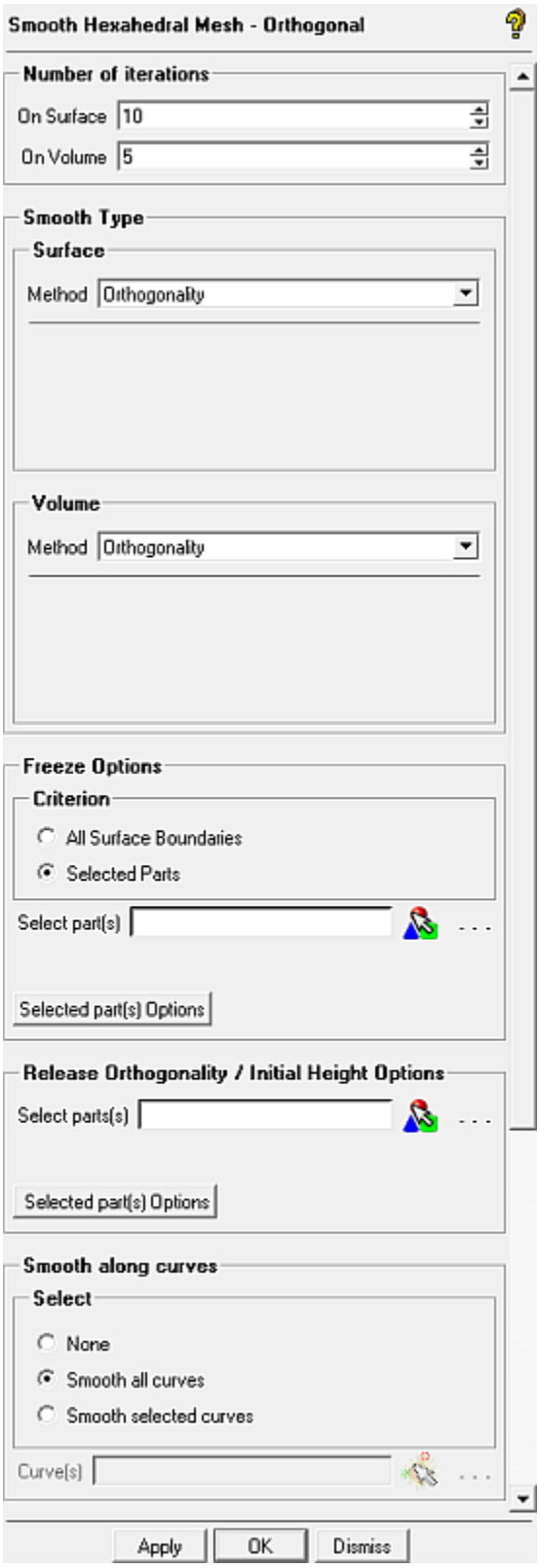
If you smooth the pre-mesh (Blocking), the smoother may move vertices and change edge distributions undesirably. If you are outputting a structured mesh, it is recommended that you save the blocking first so you can return to the previous state if necessary. If you are outputting in an unstructured mesh format, it is recommended that you first convert to unstructured mesh and then use the **Edit Mesh > Smooth Hexahedral Mesh Orthogonal** option.

Tips to improve smoothing success:

- Create the best possible starting condition by matching edge distributions.
- Create points (geometry) at the ends of the edges, and then associate the vertices (block) to them. This may be used to contain an area of fine mesh along a wall, preventing it from being totally smoothed out.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following smoothing parameters are available:



Number of iterations on Surface

specifies the number of iterations the smoother uses to relax the surface mesh. The minimum number of smoothing iterations to be completed for stability reasons is 10. The default number of iterations on surface is also 10.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of iterations on Volume

specifies the number of iterations the smoother uses to additionally relax the volume mesh after surface smoothing has been finished (minimum is 0). When the number specified is greater than 0, a volume smoothing step will be performed after each surface smoothing step to adjust the volume mesh to the surface mesh. The default number of iterations on volume is 5.

Smooth type

For both **Surface** and **Volume**, the **Method** drop down lists contains the following three options:

- **Orthogonality**

  The smoother will try to retain orthogonality and the height of the first layer. For surfaces, orthogonality implies that the first layer grid lines will be orthogonal to the surface edges.

- **Laplace**

  The smoother will try to equalize the mesh by setting the control function f to 0 in the elliptical differential equation.

- **Structured**

  The smoother offers two additional choices: **Sorenson** methods attempt to maintain node distributions (bunching) near the surface boundaries while improving orthogonality. It could be one of two types. **Hilgenstock** methods maintain orthogonality and first layer height, and may be one of two types.

  Selecting **Structured** causes a second drop down list to appear. The following four options are available:

  – **Sorenson/Laplace**: This attempts to improve orthogonality at the boundary while maintaining the first layer height from the boundary surface and making a uniform node distribution in the interior.

  – **Sorenson/Thomas & Middlecoff**: This method improves orthogonality at the boundary while maintaining the first layer height from the boundary surface and holding the original clustering on the interior.

  – **Hilgenstock/Laplace**: This attempts to improve orthogonality and uniform node distribution within the mesh.

  – The **Hilgenstock/Thomas & Middlecoff** method tries to maintain orthogonality at boundaries while holding the first layer height. With Thomas & Middlecoff as the background control function, the original clustering is intended to be maintained.

Note

During smoothing, the volume mesh will be separated into an inner (**Volume**) part and boundary (**Surface**) part which will be smoothed independently of each other. The boundary (surface) elements will be smoothed first, then the inner volume will be adjusted to the smoothed surface boundary, and finally the inner (volume) elements will be smoothed. Hence, it is advisable to set the same **Smooth type** for both **Surface** and **Volume**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

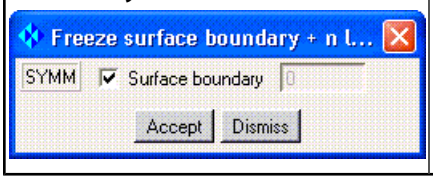
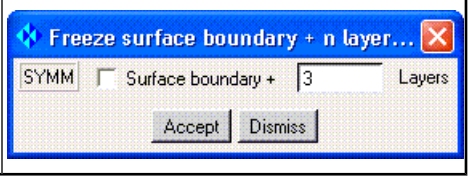
Freeze Options

- **All Surface Boundaries**

  freezes all boundary node locations.

- **Selected Parts**

  allows you to specify freeze options for selected parts. Select the part(s), then click **Selected part(s) Options**. Specify the frozen surface boundary by enabling **Surface boundary**. Specify the number of layers to be frozen with the surface boundary by disabling **Surface boundary** and entering the number of layers n in the **Layers** field. The number of layers is set to 0 by default. When **Layers** is greater than 0, then the surface boundaries including the first n layers from the boundary will be frozen. For example, a value of 3 indicates that three layers of nodes away from the surface will be frozen along with the surface boundary.

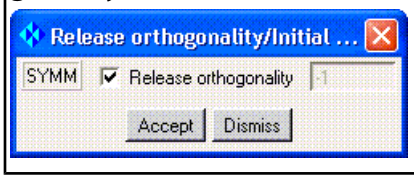
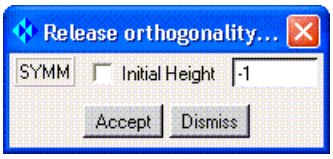
(A) Enabling **Surface boundary**	(B) Specifying number of layers
	

Release Orthogonality / Initial Height Options

For certain situations, it may be helpful to release the orthogonality requirement from a certain surface part or set the first layer distance (initial height of the first element off the wall) on certain surfaces. These are mutually exclusive since orthogonality is required to set the first layer height. Select the desired parts, and then click **Selected part(s) Options**. The default is to release orthogonality for each part. You can set the initial height by disabling **Release orthogonality** and then set the initial height by entering it in the **Initial Height** field.

Note

You can set the **Initial Height** to **-1** (default) in order to keep the starting initial height.

(A) Enabling **Release orthogonality**	(B) Specifying **Initial Height**
	

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

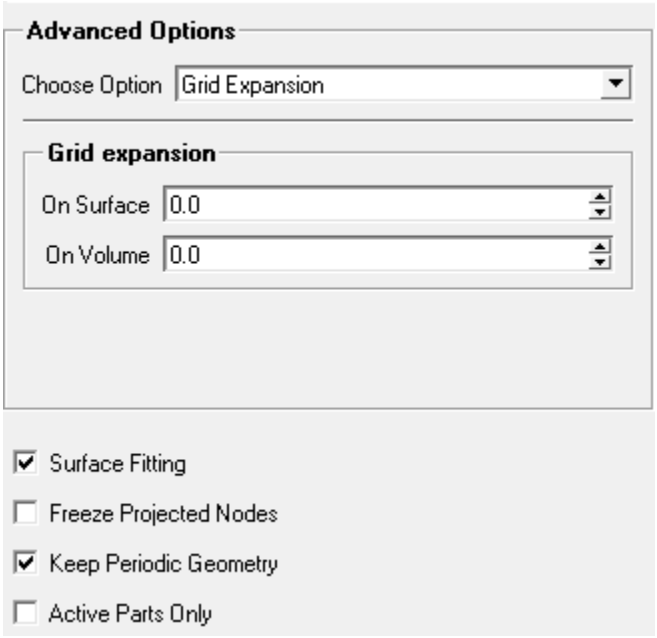
Smooth along curves

allows you to specify which edges (projected onto curves) should be smoothed. The smoothing will be carried out when faces are smoothed. You can specify none, all, or selected curves. The default is all.

Note

If the mesh distribution has a relatively large dynamic range of mesh sizes along a curve, then selecting this option may be counter productive.

Advanced Options



The **Choose Option** drop down list contains the following advanced smoothing options:

Grid Expansion

If a smoothing method other than Laplace is selected, then the grid expansion values will be used to distribute the control function values into the inner volume. If the grid expansion rate is greater than 0.0, the algorithm computes a pseudo structured region in which the control function values will be interpolated by an exponential decay (the grid expansion rate will then control the negative exponent (e power -g) of the control function values). If it is 0.0 then a Laplace interpolation of the control function values will be done, i.e., for an inner node, the control function value will be averaged by the control function values of its neighbor nodes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default values are 0.0 for the surface mesh and for the inner volume. Reasonable values other than 0.0 should be greater than 1.0.

Note

It may be difficult to form the pseudo structured regions in some cases. If you have such problems, set the grid expansion rate back to 0.

Note

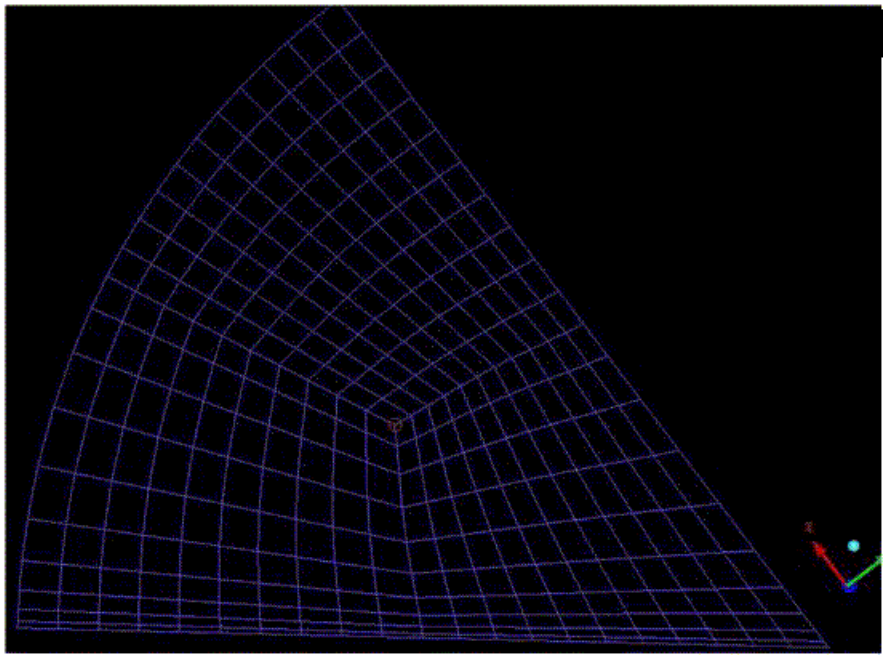
If the grid expansion rate is set to 0.0 and the smooth type is set to **Orthogonality**, the first layer height and the orthogonality at the boundary are still achieved, but the next layers tend towards Laplace approaching the middle of the mesh. In case of a Navier- Stokes mesh you can see the effect near the boundary where the first layer is orthogonal to the boundary but the next layers curve as they equalize (Laplace) the distances of the layer nodes.

Treat unstruct nodes

determines the method of treating inner unstructured nodes. An unstructured node is a node on which no ij (surface) or ijk (volume) directions can be defined, e.g. a node which has more or less than 4 (surface) or 6 (volume) neighbors is clearly an unstructured node.

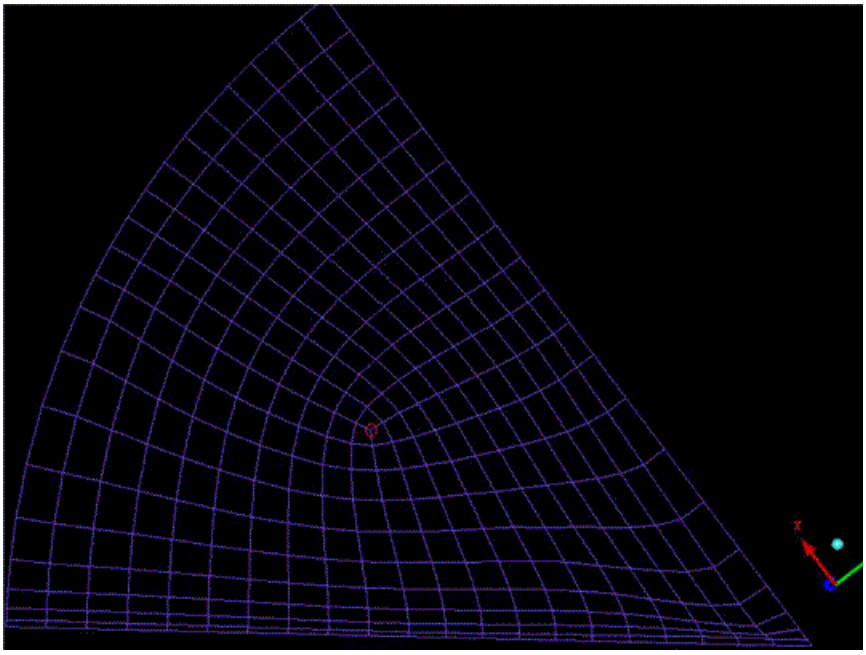
Figure 422: Initial Mesh (p. 499) shows the mesh before smoothing. The highlighted node is an unstructured node, while all other nodes are structured. Figure 423: Smoothed Mesh (p. 500) shows the mesh after smoothing.

Figure 422: Initial Mesh



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 423: Smoothed Mesh



- **Bisector**

  Each structured neighbor node will be positioned in a bisector angle with respect to the unstructured node and the two other neighbors. The grid line between the node and the unstructured node will be in a bisector angle to the 2 grid lines between the unstructured node and the corresponding neighbor node from left and right. The distance will be calculated in a heuristic way. If the neighbor node is unstructured then it will be positioned using the Laplace method.

- **Laplace**

  Each neighbor node will be positioned using the Laplace method.

- **Modified Laplace**

  Each structured neighbor node will be positioned using the Laplace method, but comparing its distance to the unstructured node with the appropriate distances of the unstructured node to its other neighbors (should not be greater than a certain ratio). If the neighbor node is unstructured then it will be positioned using the Laplace method.

The hexahedral smoother works better on structured meshes because it can use structured elliptical smoother methods which are more powerful than the methods for the unstructured parts for which it mainly uses Laplace.

Note

For most models, the effect of the this setting may be negligible.

Stabilization Factor
is used in the calculation of the new nodes and should be greater or equal to 1.0. A higher factor will make the smoother more stable at the cost of orthogonality at the boundaries. If there are problems with the smoother it is recommended to increase this value (to around 8.0). Default values

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are 1.0 on surfaces and 2.0 in the volume. Increasing this parameter may be helpful on certain model configurations if the smoother corrupts the mesh, especially if it appears due to overly orthogonal boundaries.

Use Orthogonal Distance
if enabled, then the original boundary distance of a first layer node will be calculated by projecting it to the boundary and measuring the distance to the projected point. Otherwise, the length of the original grid line will be used. By default this option is disabled for both the surface and volume.

In cases where there is a sharp angle between the grid lines from the first layer to the boundary, if **Use Orthogonal Distance** has not been set, the calculated first layer distance may be considerably greater than the distance of the first layer nodes to the boundary (because the length of the grid line will be used). Hence, it may be advisable to set **Use Orthogonal Distance** in the same way for both **Surface** and **Volume**. This would ensure that the first layer grid line angles in the volume near the surface boundary would be similar to the nearby surface boundary grid line angles.

Define Edges on Part Border Options
The border grid lines attached to the selected parts will be defined as edges. The main purpose for defining these edges is that they separate the two sides of the surface mesh. If an edge is topologically internal then orthogonality will not be established there.

Fix Orientations
If **Volume** is enabled, then before smoothing takes place, all volume elements will be checked to see if the node order defines a right-handed element and will be fixed if necessary.

Note

Enabling this option could result in additional calculation time.

Surface Fitting
constrains boundary nodes to the true geometry surfaces. When disabled, the original mesh faces are used to determine the boundary constraints. This option is enabled by default.

Freeze Projected Nodes
When set, all nodes which are prescribed or projected to the geometry (curves or surfaces) will not be allowed to move.

Keep Periodic Geometry
when enabled, ensures that the mesh will stay on the geometry on periodic sides. Otherwise the periodic nodes can move away from the geometry. Nevertheless periodicity is still guaranteed. This option is enabled by default.

Active Parts Only
if enabled, only the active parts (parts that are turned on (activated) in the **Parts** branch of the display tree, see the Parts (p. 180) section) will be smoothed. The geometry or mesh of the active parts does not need to be displayed. When this option is disabled, the whole mesh will be smoothed. The histogram displayed will also reflect the setting of the **Active parts only** option. This option is disabled by default.

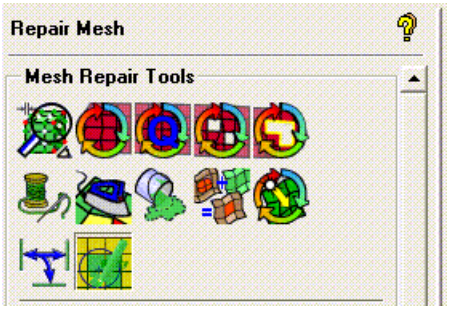
If you click **Apply** or **OK** the mesh smoothing will be started. If **OK** is selected, then the window will be closed afterwards.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repair Mesh

The **Repair Mesh** options can be used for manually editing parts of the mesh that are not of good quality.

Figure 424: Repair Mesh Options



The following mesh repair options are available:

Build Mesh Topology
Remesh Elements
Remesh Bad Elements
Find/Close Holes in Mesh
Mesh From Edges
Stitch Edges
Smooth Surface Mesh
Flood Fill / Make Consistent
Associate Mesh With Geometry
Enforce Node, Remesh
Make/Remove Periodic
Mark Enclosed Elements

Build Mesh Topology

Angle

Line elements (1D) will be created between shell elements (2D) attached at angles greater than the specified angle. Node elements (0D) will be created between line elements (1D) attached at angles greater than the specified angle.

Create bars between shell parts

automatically creates bar elements between different shell mesh parts, regardless of the angle between them.

Selected elements

allows you to select particular elements for which to build mesh topology.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remesh Elements

Only one element type can be remeshed at a time. Select a particular surface mesh type from the **Mesh type** drop-down list and then manually select particular elements. The selected elements are first deleted, and then the vacant area is remeshed. The element sizes are determined by the size of the surrounding edges.

Note

This option is designed for plane subsets of the surface mesh, and for selected volume parts, but not for curved surfaces or entire volumes.

Part
The remeshed elements will be added to the selected part.

Surface projection
allows you to project new nodes onto the nearest surface.

Ignore projection
if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore projection** is disabled, the elements will not be able to be remeshed as the projections will be lost.

Note

It is recommended to enable **Ignore projection** when remeshing a controlled region of mesh, and disable it when remeshing bad elements for the entire model.

Number of offset layers
specifies the total number of offset layers while remeshing the selected elements.

Offset height factor
The height of the first offset layer is this factor multiplied by the given element size. The height of any successive layers will increase for concave edges, and decrease for convex edges.

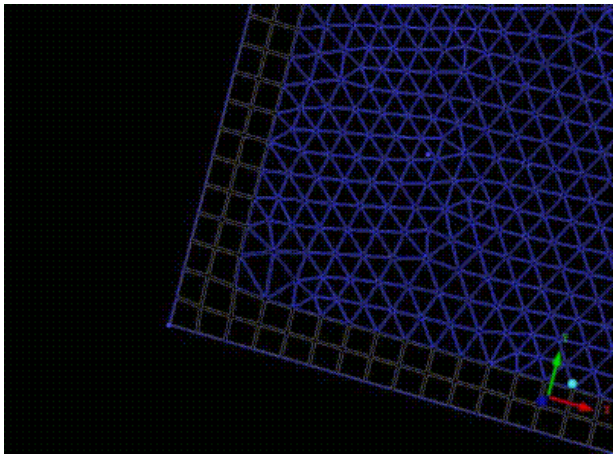
Offset type

- **Standard**

  Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front may not be identical as the number of nodes on the initial boundary.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

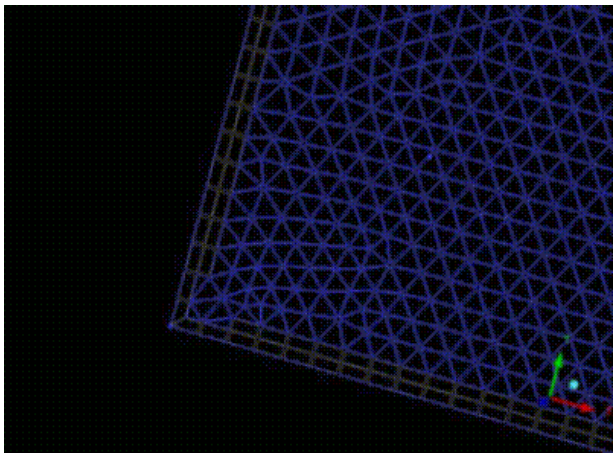
Figure 425: Standard Offset Example



- **Simple**

  Offsets will be created normal to the edges without special solutions for sections with small or large angles, such as corners. The number of nodes on the offset front is identical to the number of nodes on the initial boundary.

Figure 426: Simple Offset Example

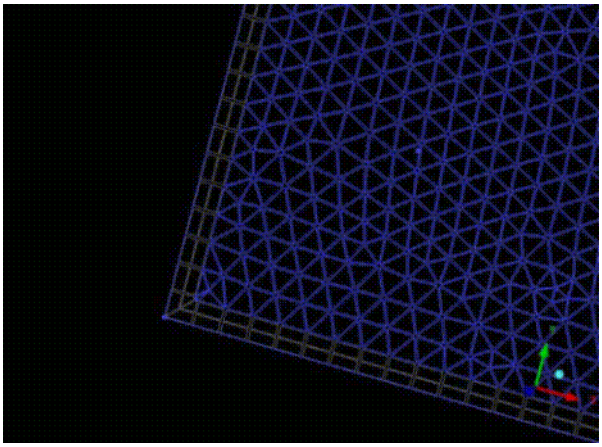


- **Forced Simple**

  This option is the same as Simple Offset, but without collision checking.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 427: Forced Simple Offset Example



Remesh Bad Elements

The **Remesh Bad Elements** option has similar options to Remesh Elements (p. 503), with the difference that only the bad quality elements from the selected elements will be deleted and remeshed. It will not delete all selected elements.

Mesh type
specifies the mesh type.

Elements
specifies the elements to remesh.

Surface projection
allows you to project new nodes onto the nearest surface.

Ignore projection
if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore projection** is disabled, the elements will not be able to be remeshed as the projections will be lost.

Note

It is recommended to enable **Ignore Projection** when remeshing a controlled region of mesh, and to disable it when remeshing bad elements for the entire model.

Quality metric
specifies a quality metric.

Up to quality
specifies the desired quality level.

Max iterations
is the total number of iterations to remesh in order to get the desired quality level.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Find/Close Holes in Mesh

The **Find/Close Holes** option locates holes in the selected elements and remeshes them.

Mesh Type
specifies the appropriate surface mesh element type.

Elements
specifies the elements from which to look for holes.

Surface projection
allows you to project new nodes onto the nearest surface.

Part from geometry
when enabled, the Part for the new elements is inherited from the adjacent surface part. This is calculated with a normal ray from the center of each new element to the nearest surface. When this option is disabled, the Part for the new elements is inherited from the surrounding elements based on the first element selected.

Mesh From Edges

The **Mesh from Edges** option allows you to close holes by selecting the edges that surround it.

Mesh type
specifies the appropriate surface mesh elements available.

Edges
allows you to select the edges that surround the holes.

Surface Projection
allows you to project new nodes onto the nearest surface.

Single loop & complete edges
if enabled, it will try to complete a loop from the edges selected. If disabled, it will not mesh if the selected edges do not form a complete loop.

Interpolation surface
interpolates the meshing area by a biquadratic approximation in order to generate a smooth surface.

Keep volume consistent
if enabled, makes sure that the surface mesh and volume mesh are consistent with each other after remeshing.

> **Note**
>
> This only applies to tetra meshes.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stitch Edges

The **Stitch Edges** option allows you to close gaps within selected edges (usually single edges) by merging opposing nodes, making the mesh on both sides conformal.

Edges
specifies the edges to be stitched.

Rel. Merge Tolerance
if two nodes are within this tolerance, they will be merged instead of stitched together. Specify a factor (from 0 to 1) of the local element size. Typical values are 0.05 – 0.2. The purpose of this tolerance is to avoid tiny element edges, not to limit the stitch area.

Merge End Nodes
when enabled, this option will merge end nodes when stitching the selected edges.

Smooth Surface Mesh

The **Smooth Surface Mesh** option is an automatic process for improving the quality of selected surface mesh elements. A common mesh smoothing technique is Laplace Smoothing, which seeks to reposition the nodes so that each internal node is at the centroid of the polygon formed by its connected neighbors. This repositioning is usually done iteratively.

Elements
specifies the surface mesh elements to smooth.

Max Iterations
is the maximum number of iterations for smoothing the mesh.

Accuracy
is a relative tolerance for element quality (between 0 and 1) related to the local element size. The default value is 0.025. If set to 0, then the given number of smoothing steps is done. Bigger values improve the performance because smoothing stops when the accuracy has been reached.

Isoparametric
is an additional option for the surface smoother. The default value is 0, which is pure Laplacian smoothing. You can increase this parameter up to 1 to produce an isoparametric mesh.

Surface projection
allows you to project new nodes onto the nearest surface.

Smooth boundaries
smoothes the boundary nodes of the surface mesh selected, not just the internal nodes. It also allows curve nodes to be improved.

Flood Fill / Make Consistent

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flood Fill

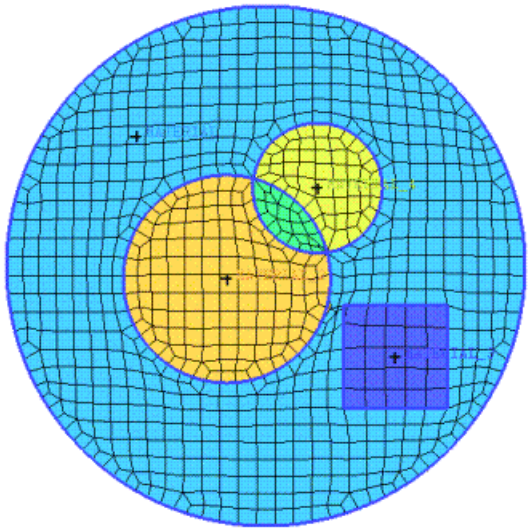
The Flood Fill process here is the same as what happens during the Octree meshing process. Flood Fill uses Material points to mark volume elements and group them into volume parts. See also the Octree meshing approach in the **ANSYS ICEM CFD User's Manual**.

For each material point in the model, a root volume element is defined (the volume element that this material point sits in). From each root volume element, all the neighboring volume elements will be added, and then their neighbors are added in a flood filling manner. This process stops at the surface elements which define the closed boundary and selects all the volume elements within the boundary. The selected volume elements are assigned to the part of the material point within the volume.

Note

If the flood fill process finds a second material point with a different part name within the same process, leakage is detected. Flood fill will attempt to automatically close holes, but if it cannot, you will be prompted to close holes interactively. A message will alert you to which material point entities are in conflict. In the interactive process, you will be shown a jagged line representing the connection path between the material points along the centroids of the volume elements and through the "hole" if there is one. It will also highlight the single edges and bring up the mesh editing option for mesh from edges so you can more easily select the edges and close the hole. Then the flood fill will resume. In some cases, the problem may be that both material points are in one volume or perhaps one is in the wall between two volumes; in which case deleting or moving material points is the solution.

Figure 428: Use of the Flood Fill Option



Note

For situations where the mesh is not surface fitted, use the **Mark Enclosed Elements** option instead of **Flood Fill**.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make volume mesh consistent with surface mesh

makes tetra volume mesh consistent with tri surface mesh. Tetra elements near the surface are modified to fit the surface mesh. The final volume mesh is connected node for node with the surface mesh. You can use this command to replace a boundary or cut a surface mesh out of a volume.

Figure 429: Original Volume and Surface Meshes (p. 509) shows a manifold with an inserted baffle. The tetra volume mesh for the manifold and the tri surface mesh for the baffle is also shown.

Figure 429: Original Volume and Surface Meshes

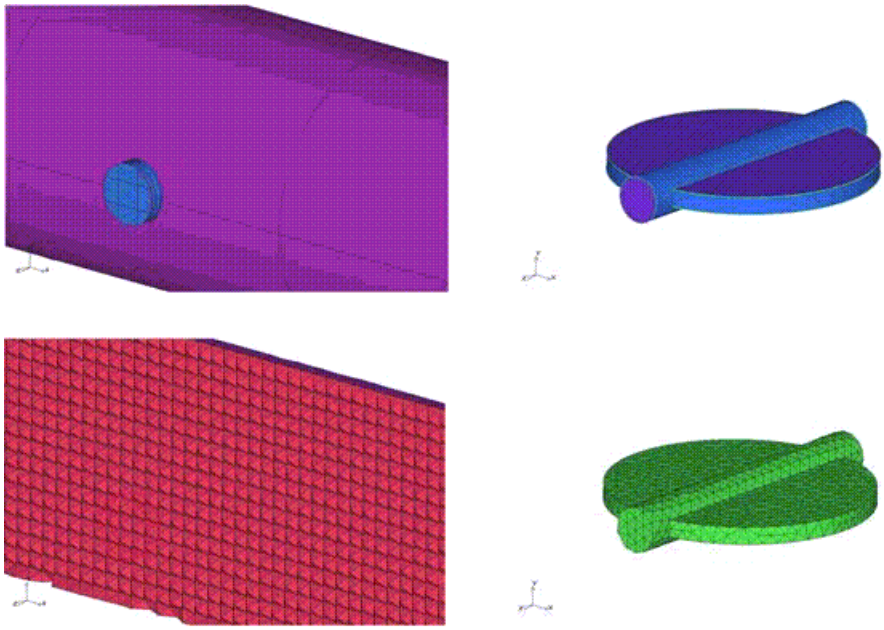
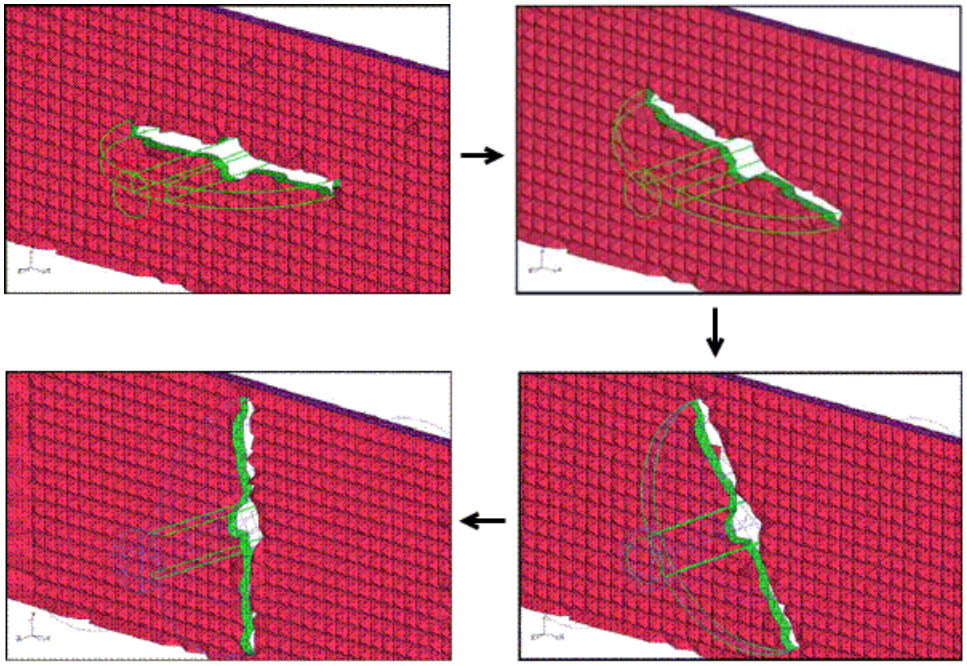


Figure 430: Combined Mesh (p. 509) shows the combined mesh, where the use of this option allows you to combine the two meshes in different orientations.

Figure 430: Combined Mesh



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option may be chosen for **All** elements, or **Selected** elements.

Note

Most manual mesh editing commands, such as merge, split, etc. automatically keep the volume consistent. Other commands, such as remesh from edges have an option to "Keep volume consistent".

Associate Mesh With Geometry

The **Associate Mesh with Geometry** option will assign the closest part to the selected surface mesh elements.

Enforce Node, Remesh

If a node lies outside the mesh and is not the part of mesh (e.g., a Weld node) the **Enforce Node, Remesh** option will remesh nearby surface nodes so that the mesh respects the node location.

Mesh type
specifies the mesh type.

Surface projection
allows you to project moved nodes onto the nearest surface.

Ignore projection
if enabled, projections to curves and points will be ignored when remeshing. If a group of elements that are being remeshed have internal nodes that are projected to curves or points, and if **Ignore projection** is disabled, the elements will not be able to be remeshed as the projections will be lost.

Note

It is recommended to enable **Ignore projection** when remeshing a controlled region of mesh, and to disable it when remeshing bad elements for the entire model.

Keep volume consistent
if enabled, then the volume mesh also gets remeshed with the shell mesh, and the consistency between the volume and surface mesh is maintained.

Note

This option only applies to tetra meshes.

Selection Method
You have the option of selecting nodes or points. If selecting points, also specify the surface mesh part name for the new node element.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make/Remove Periodic

The **Make/Remove Periodic** option allows you to manipulate the periodicity of specific nodes. You can check the existing periodicity through the mesh display options in the Model tree.

Note

In order to make nodes periodic, periodicity must be defined in Mesh > Global Mesh Parameters.

Make Periodic
allows you to select two nodes at a time to define periodic node pairs.

Note

In order to make an axis vertex periodic with itself, select only that node, and you will be asked to confirm that the selected node should be made periodic to itself.

Remove Periodic
removes the periodicity from the selected nodes. Select one node of each periodic node pair.

Mark Enclosed Elements

The **Mark Enclosed Elements** option uses Material Points within an enclosed volume of surfaces to mark all the elements within that volume and assign them to the same Volume Part as the enclosed Material Point. Only mesh in regions containing the selected Material Points will be affected.

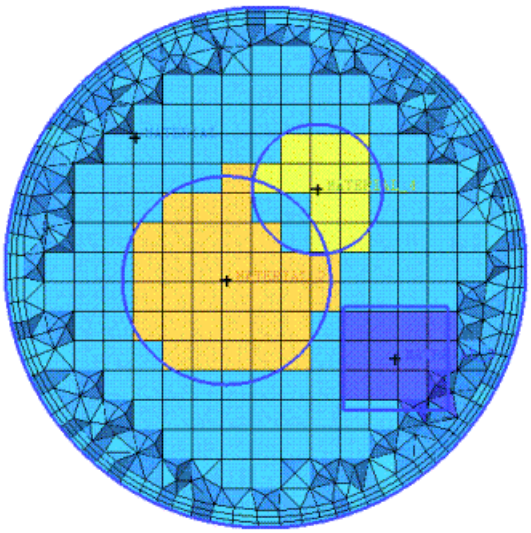
Any elements within the enclosed space containing the Material Point are placed into the same Volume Part. If the Material Point is outside of the selected surfaces, then all elements with centroids outside of the enclosed surfaces are placed within the Volume Part. If there is an enclosed volume that does not contain any material points, its elements are not affected.

The end result is similar to that obtained using Flood Fill, but the approach is less restricted because the mesh does not need shells or faces aligned with the enclosing surfaces. For most situations, Flood Fill is a more robust and faster method. For situations where the mesh is not surface fitted, this option should be used in place Flood Fill.

In Figure 431: Example of the Mark Enclosed Elements Option (p. 512), note that the elements are marked based on the location of the centroid of each element. Note also that the region between the cylinders is an enclosed volume but has no Material Point, so its elements remain with the original material.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 431: Example of the Mark Enclosed Elements Option



Enclosed Surfaces
specifies the enclosed surfaces that form a volume.

Material Points / Bodies
specifies the Material Points.

Only volume elements
allows you to assign only volume elements within the enclosed surfaces to the part of the Material Point. When this option is disabled, shell elements also within the enclosed surfaces will be assigned to the part of the Material Point as well.

Note

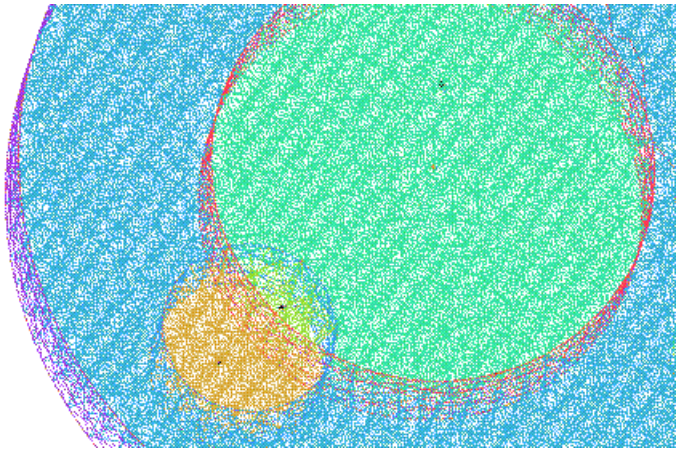
This option is enabled by default as most solvers prefer to keep volume elements and shell elements in different parts.

An example is shown in Figure 432: Using the **Only Volume Elements** Option (p. 512). When the **Only volume elements** option is enabled, the volume elements within the enclosed surfaces are included in the Material Point Part. When the **Only volume elements** option is disabled, the surface elements within the enclosed surfaces are also included in the Material Point Part.

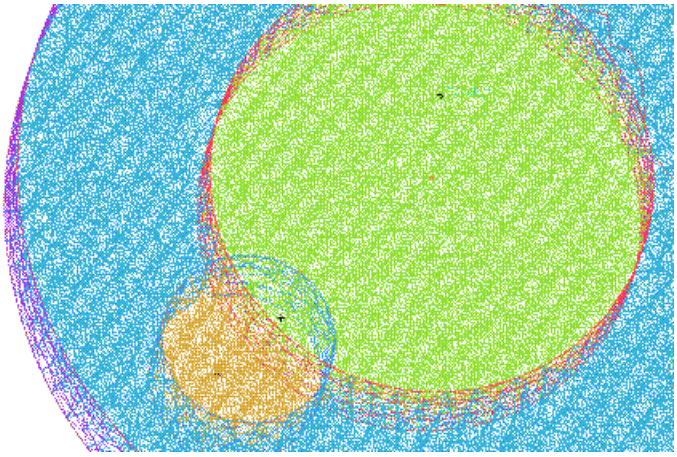
Figure 432: Using the Only Volume Elements Option

Original Mesh

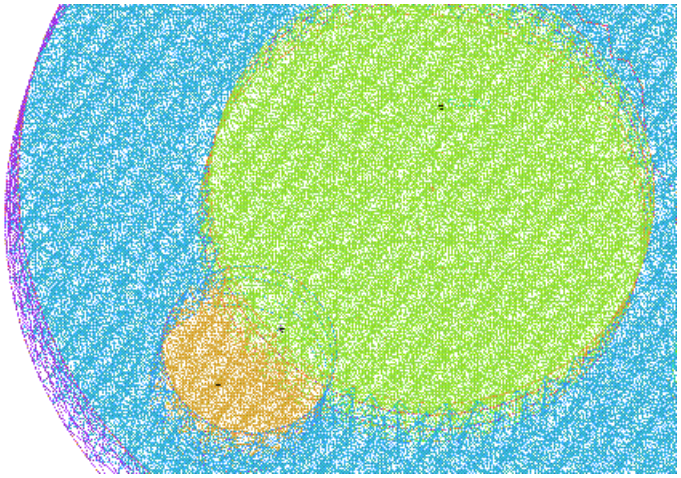
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Only volume elements Enabled



Only volume elements Disabled

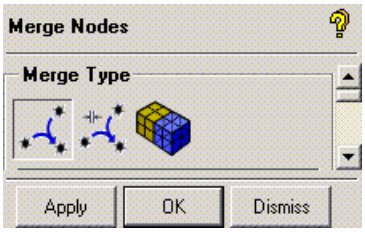


Merge Nodes

The **Merge Nodes** option allows you to merge two nodes together to improve mesh quality while editing the mesh. Also, disconnected parts of mesh can be merged together.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 433: Merge Nodes Options



The following options are available for merging nodes:

Merge Interactive
Merge Tolerance
Merge Meshes

Merge Interactive

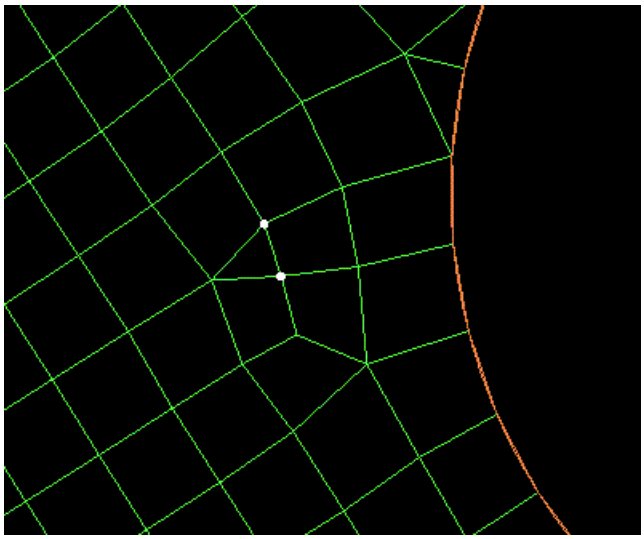
The **Merge Interactive** option allows you to merge the selected nodes.

Nodes

specifies the nodes to be merged.

Figure 434: Selection of Nodes to be Merged (p. 514) shows two nodes to be merged.

Figure 434: Selection of Nodes to be Merged

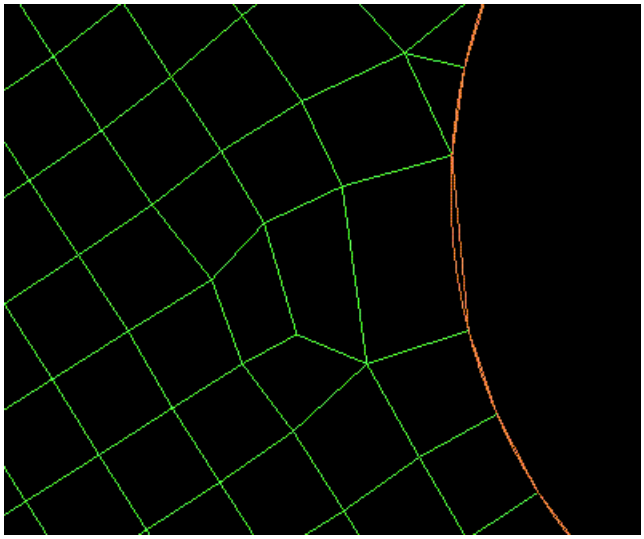


Propagate Merge

propagates the merged node through the mesh until propagation is stopped by a tri element or a mesh boundary. Figure 435: Propagate Merge (p. 515) shows the use of the **Propagate merge** option.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

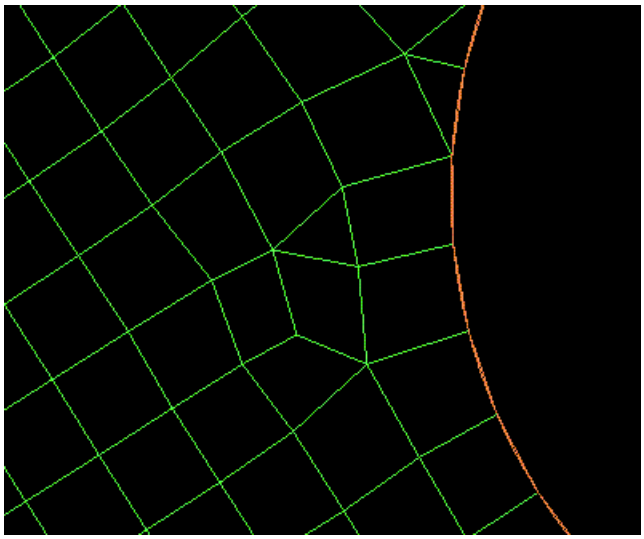
Figure 435: Propagate Merge



Merge to average

only the nodes selected will be merged. Element types will be changed if necessary to perform the merge. Using the same example as Figure 434: Selection of Nodes to be Merged (p. 514), Figure 436: Merge to Average (p. 515) illustrates this operation.

Figure 436: Merge to Average



If the mesh is only surface mesh, you will be given the option to terminate or propagate the merged nodes after selecting the nodes. After making the selection, that method will be used on all following nodes merged until the function is exited by using the middle mouse button.

Ignore projection

ignores restrictions based on node projection during merging. For instance, two point projected nodes cannot be merged because they are each confined to their respective points. This option ignores that restriction and allows the merge.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge Tolerance

The **Merge Tolerance** option allows you to merge nodes that are within a specified tolerance.

Nodes
specifies the nodes which are checked for proximity within the tolerance.

Tolerance
specifies the tolerance value. This function will attempt to merge pairs of selected nodes whose proximity to each other is within the tolerance.

Ignore Projection
ignores restrictions based on node projection during merging. For instance, two point projected nodes cannot be merged because they are each confined to their respective points. This option ignores that restriction and allows the merge.

Only on Single Edges
allows you to easily select all the mesh, but will only perform node merging on the perimeter of the surface mesh (single edges). Interior nodes, those on double or multiple edges, are not merged with this option, even if their proximity is within the tolerance.

Note

To color-code single edges yellow, right-click on **Shells > Diagnostics** in the Display tree and enable **Single edges**.

Merge Meshes

The **Merge Meshes** option allows you to merge disconnected parts of the mesh if two or more domains are loaded and the nodes do not match up at the interface between them. These domains are usually created separately. The merging will match up the nodes so that one-to-one connectivity is maintained throughout the entire merged domain. Tri elements at the interface of one domain should be no larger than roughly 3 times the tri element size of the other domain. The larger elements will be subdivided at the interface until they reach the size of the elements of the other domain. Tetra/tetra and tetra/hexa merges can also be done. If a tetra mesh and hexa mesh are to be merged, pyramids will be created at the interface.

Note

The **Merge Meshes** option only works well when the quad elements at the interface are close to equilateral.

The following methods are available for merging the mesh:

Merge volume meshes
is used to merge two different volume meshes (both meshes can be tetra or one can be tetra and the other hexa).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge surface mesh parts

allows you to specify the surface mesh parts to be merged. Select the surface mesh parts at the interface of the two separate domains. Only one part can be selected if the surface elements of the two separate domains are in same part at the interface.

Frozen volume mesh parts

specifies the volume parts to remain fixed. The volume mesh part that is not frozen will be modified. If one of the volume parts is hexa mesh, then that part will be frozen whether it is selected as frozen or not.

Resolve refinements

is used to resolve couplings, which define the connectivity between elements which are not connected node by node (hanging nodes). In order to accommodate solvers that cannot handle couplings, this method can be used to resolve couplings using refinement and to make the mesh more conformal.

Note

Concave regions that have a coarsened interior will be filled with fine mesh. This results in an interior refinement area with finer mesh sizes and coarse regions around it.

Standard

is the same as Pure 3D Refinement, which resolves refinements into all three directions. But this option has smarter handling of 2.5D (surface mesh that has been extruded to become volume mesh) cases.

Allow unstable patterns

allows non-standard refinement patterns. This improves the situation of heavy propagation of fine elements.

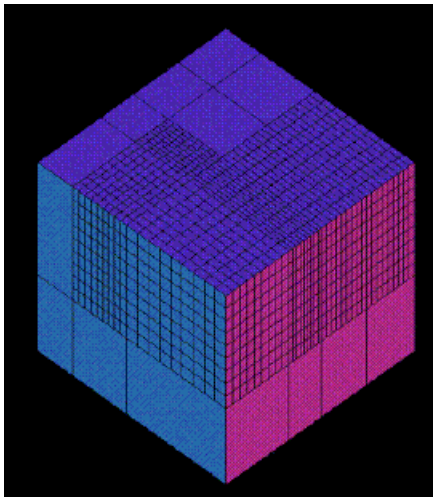
Pure 3D refinement

resolves refinements into all three directions. This may lead to heavy propagation of fine elements.

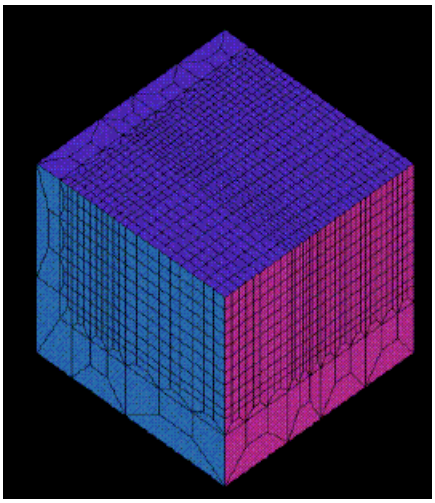
See Figure 437: Resolve Refinements Options-Example 1 (p. 517) and Figure 438: Resolve Refinements Options-Example 2 (p. 518) for some examples of the different options.

Figure 437: Resolve Refinements Options-Example 1

Original Mesh



Standard and **Pure 3D Refinement** results in propagation of fine mesh in concave regions.



Allow Unstable Patterns

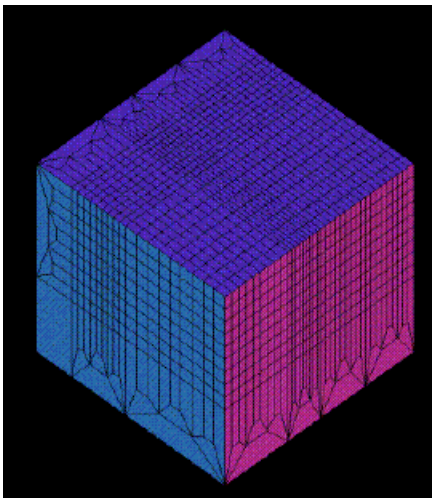
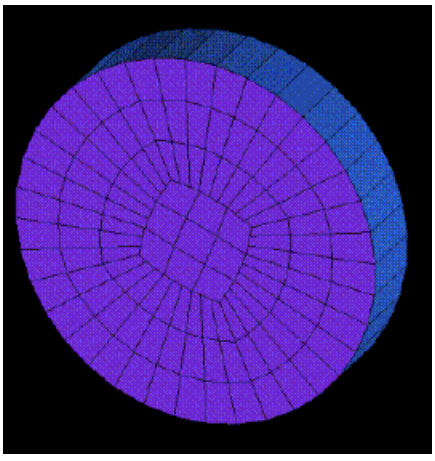


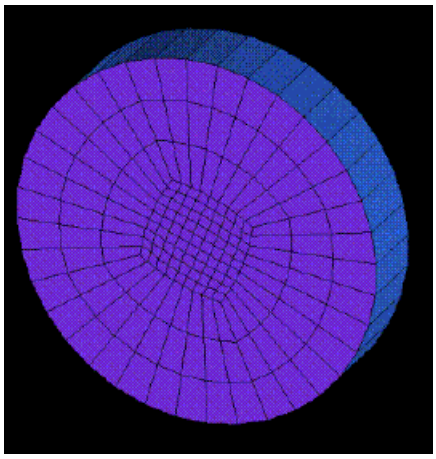
Figure 438: Resolve Refinements Options-Example 2

Original Mesh

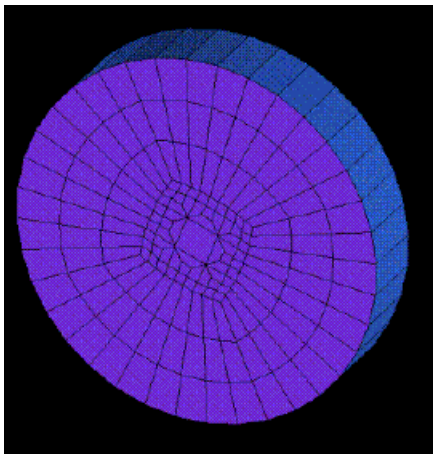


Pure 3D Refinement

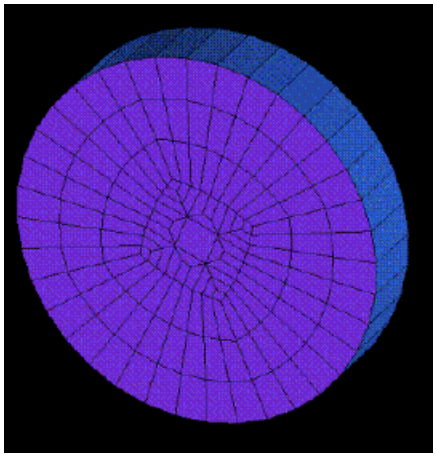
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Standard



Allow Unstable Patterns

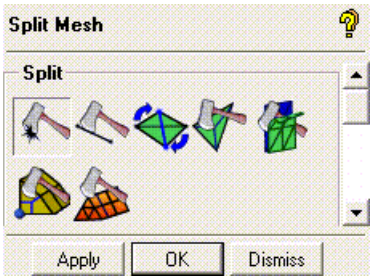


Split Mesh

The **Split Mesh** option allows you to split mesh elements at an individual level. This is one type of refinement of elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 439: Split Mesh Options



The different options available under this menu are as follows:

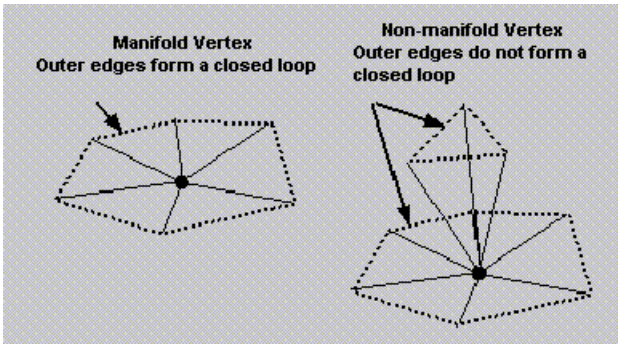
Split Nodes
Split Edges
Swap Edges
Split Tri Elements
Split Internal Wall
Y-Split Hexas at Vertex
Split Prisms

Split Nodes

The **Split Nodes** option allows you to split nodes for triangle mesh and move them. Both nodes will have the same constraint (point, curve or surface) as the original node. You need to specify whether the vertex is a manifold or non-manifold one.

Non-manifold vertices are those where the outer edges of their adjacent elements do not form a closed loop. This usually indicates elements that jump from one surface to another, forming a "tent-like" structure. This would usually pose no problem for mesh quality but will represent a barrier in the free domain that probably should not be there.

Figure 440: Manifold and Non-Manifold Vertices



Manifold

For manifold vertices, select the node with the left mouse button and accept the selection with the middle mouse button. Proceed to move the selected nodes with the left mouse button. New triangles will be created that are attached to the new node.

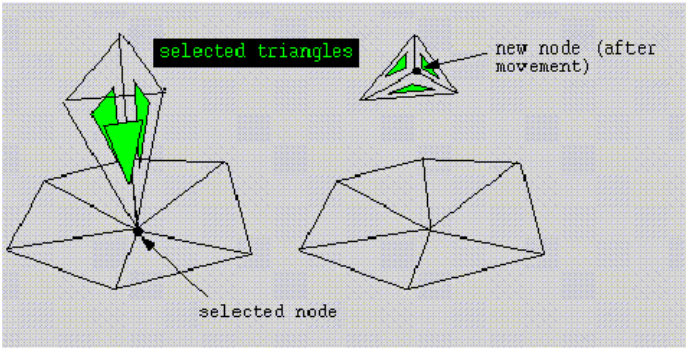
Non-manifold

For non-manifold vertices, select the node with the left mouse button. All surface elements connected to that node will then be highlighted. Select the surface elements that will be attached to the new node,

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and press the middle mouse key to accept. Then move the node with the left mouse button. No new triangles will be created.

Figure 441: Split Node of Non-Manifold Vertex



Split Edges

The **Split Edges** option allows you to split selected edges into two, as well as the adjacent elements.

The following methods are available:

Selected
allows you to split the selected edges.

For this method, the following options also apply:

Propagate
The split edge operation will propagate through the mesh until the propagation is stopped by a tri element or it exits to the ORFN region.

Project
projects the newly created nodes onto the geometry.

Split Ratio
specifies a factor between 0 and 1 to determine the location of the split along the edge.

Note

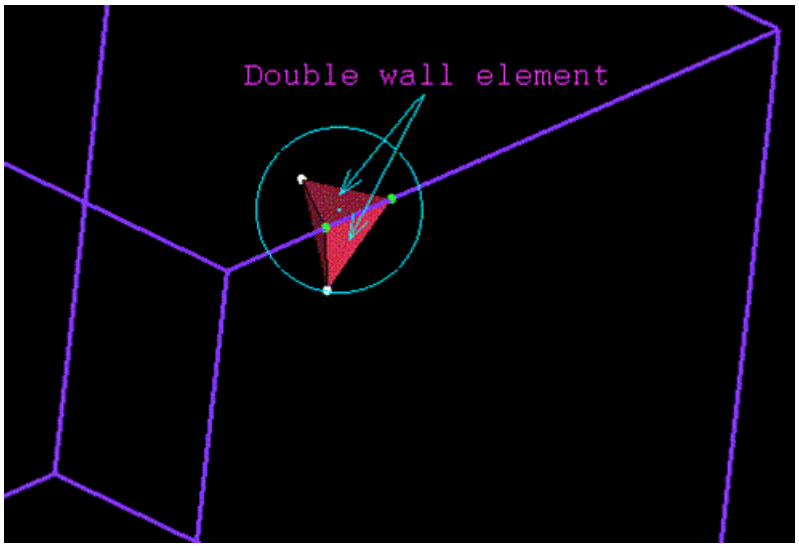
This option is available only for Hexa mesh.

Double walls
A double wall element is a tetra element that has all four nodes on the surface (two triangle faces on the surface). The fix is to split the edge spanning the volume so each tetra element has only one face on the surface. See the example shown below.

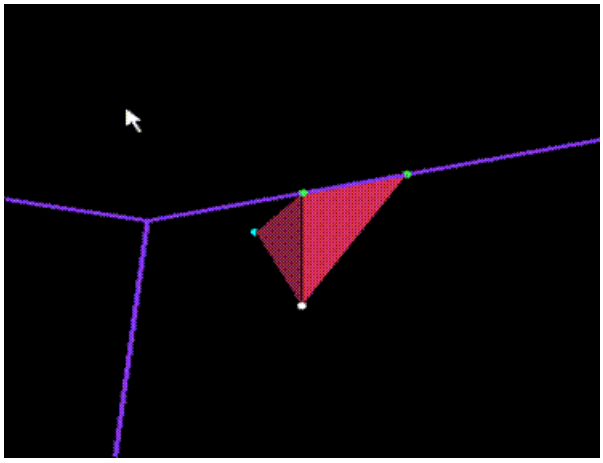
Figure 442: Double Wall Element

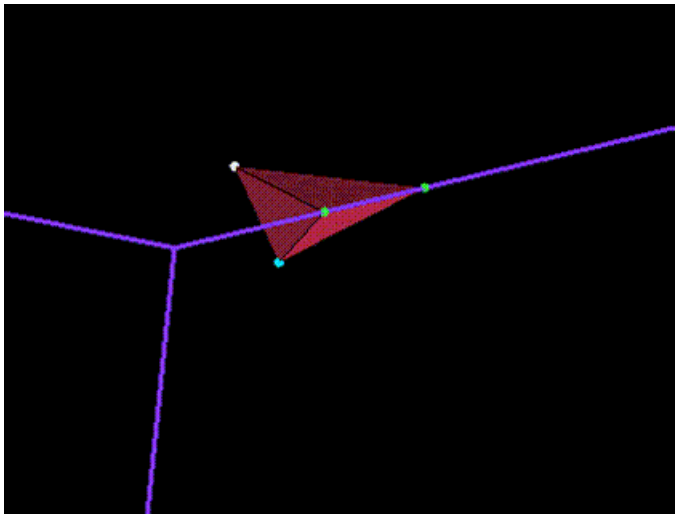
Double Wall Element

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Double Wall Element Split Into Two Elements





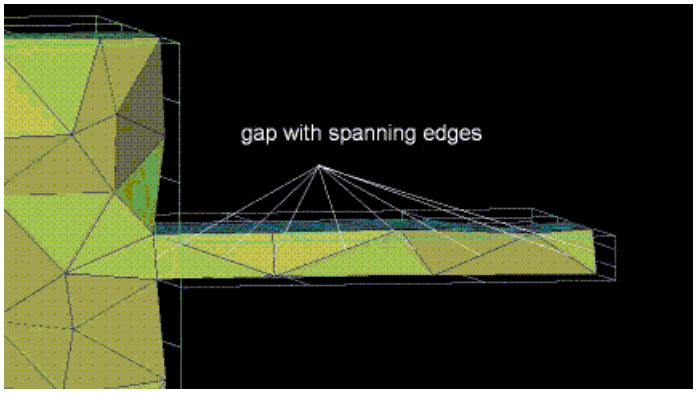
Interior spanning

If a tetra element that is located in a gap of the geometry has its nodes on two different surfaces, the analysis of that element may not yield sufficient results. This option splits the edges of all elements located between the different surfaces and are not connected to a triangle element (i.e., all edges that are not boundary edges). This will create an additional node on the edge, which will provide another face for analysis and a more accurate result. See Figure 443: Gap with Spanning Edges (p. 523) for an example.

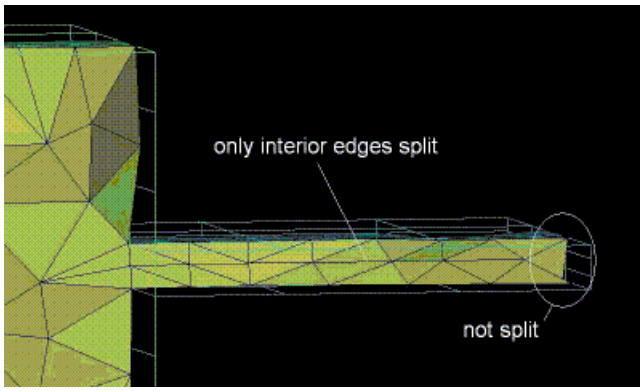
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 443: Gap with Spanning Edges

Gap with Spanning Edges

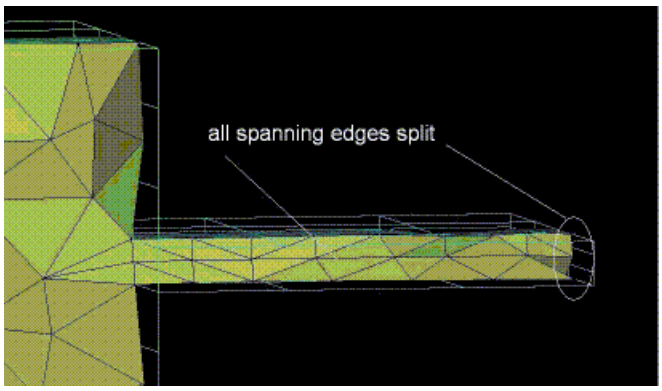


Interior Spanning Edges Split



All spanning
is the same as **Interior spanning**, except that all edges, including boundary edges, will be split.

Figure 444: All Spanning Edges Split

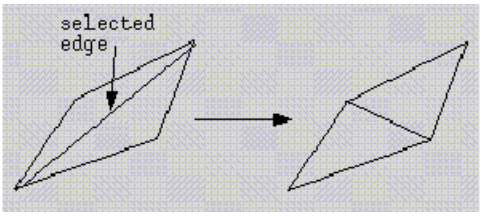


Swap Edges

The **Swap Edges** option swaps edges of two adjacent triangles. The original edge will be replaced by an edge that connects the other two corners of the triangles, as shown in Figure 445: Swap Edges Example (p. 524)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 445: Swap Edges Example



Interactive Method

allows you to select the edge to be swapped.

Automatic Method

There are two options for the Automatic method:

- By Quality

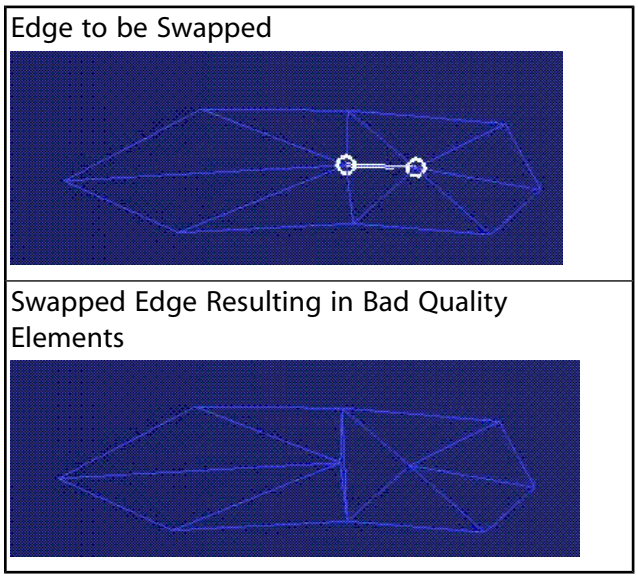
  This swaps edges to improve the mesh quality to the specified minimum quality with the specified number of iterations. The quality metric that is used is Triangle Aspect Ratio.

- By Deviation

  This swaps edges to minimize surface deviation to the specified maximum deviation with the specified number of iterations. Quality is taken into account while swapping, and edges may be swapped to improve the quality, provided it does not make the deviation worse. Also the quality of the mesh may restrict some edges from being swapped even if the deviation would be improved.

  In Figure 446: Swap Edges by Deviation (p. 524), the highlighted edge could be swapped to improve the deviation value. However, because the resulting triangles are of poor quality, the edge will not be swapped by the automatic algorithm.

  Figure 446: Swap Edges by Deviation



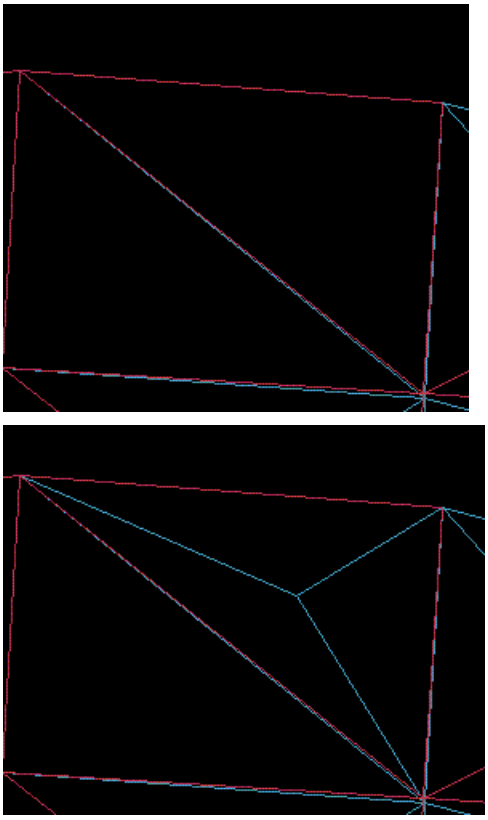
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Split Tri Elements

The **Split Tri Elements** option allows you to split selected tri elements into three triangles. The split location is at the centroid of the element.

An example of **Split Element** is shown in Figure 447: Split Element Example (p. 525)

Figure 447: Split Element Example



Split Internal Wall

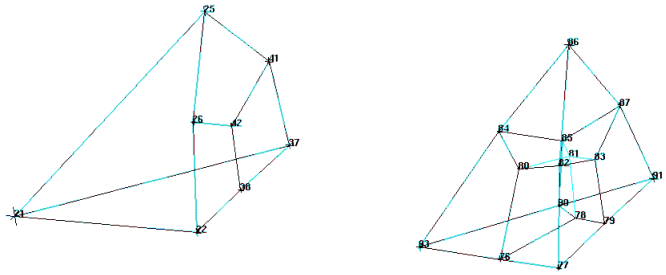
The **Split Internal Wall** option creates pairs of coincident triangles and nodes on the internal wall. Thus, the internal wall is logically treated as an actual wall with a "gap" in between it and the outer wall. This option is applicable for Tetra mesh only.

Y-Split Hexas at Vertex

The **Y-Split Hexas at Vertex** option splits hexa elements at the vertex in a Y-Grid fashion. This can be used to split degenerate unstructured hexas of pyramid or prism shape to improve the quality as shown in the example below. Only elements connected at the vertex will be affected. This can be used for surface mesh as well.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 448: Y-Split Hexas at Vertex Example



Split Prisms

The **Split Prisms** option allows you to create a few prism layers and then split them. This method is faster and can be more robust than growing the same total number of prism layers using the **Mesh Prism** option.

Selected Prism Surface Parts

specifies the selected prism surface parts. Clicking the icon will open a window with a list of prism surface parts. The prism layers on the selected surface parts will be split. If no shell mesh part is selected, then all surface prism parts will be automatically selected.

Selected Prism Volume Parts

specifies the selected prism volume parts. Clicking the icon will open a window with a list of prism volume parts. If volume parts are selected, only the prism layers of the selected surface parts that belong to these volumes will be split. If no volume parts are selected, the prism layers for all the selected surface parts will be split regardless of the volume part.

Method

specifies the method for splitting prism layers.

Fix ratio

allows you to split a prism layer such that the specified **Prism ratio** is applied to its resulting layers.

Fix initial height

allows you to split the first layer such that its first sub-layer is the specified **Initial layer height**.

Number of layers

specifies the number of layers to result from each existing layer.

Split only specified layers

allows you to split only specific prism layers.

Layer numbers (0,1,2...)

specifies which prism layers are to be split. The first prism layer is 0, the second layer is 1, etc.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

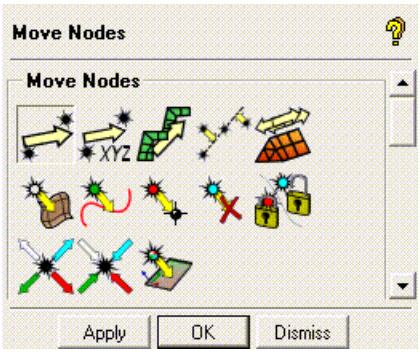
Do not split attached pyramids
when enabled, the pyramidal elements that are attached to the prism layers will not be split. This option is disabled by default.

> **Note**
>
> Using this option will result in hanging nodes in the mesh because multiple prism side faces will be covered by one pyramid's base face. Subsequent mesh checks will find uncovered faces and surface orientation errors. Most solvers cannot accept these hanging-nodes, others such as ANSYS Fluent prefer pyramids this way because they will have better aspect ratios than if they were split thin along with the adjacent prisms.

Move Nodes

The **Move Nodes** options are used to move nodes using various methods. Nodes projected to a prescribed point will not be able to be moved. Nodes that are projected to a curve will be constrained to active curves. Nodes projected to a surface will be constrained to active surfaces. Internal volumetric nodes can move along the plane defined by the screen. The different options for moving nodes are shown below.

Figure 449: Move Nodes Options



Interactive

The **Interactive** option allows you to select the node with the left mouse button and move it interactively.

Move Nodes Type
You can select **Single** to move single nodes, or **Multiple**, to move multiple nodes simultaneously.

Select
specifies the nodes to be moved.

Allow Inversion
if enabled, node movement will be allowed to invert or twist elements. If disabled, node movement will be constrained so that elements cannot be twisted.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project to

- **All geometry**

  projects nodes to the nearest geometry entities.

- **Active Parts**

  project the nodes of active parts only. Active parts include those that are enabled in the Model tree but not visible on the screen.

- **None**

  nodes will not be projected.

Exact

The **Exact** option allows you to modify the coordinates of selected nodes. The nodes may be moved individually or in relation to a reference location.

Multiple nodes can be selected and modified at the same time. Typically, this function is used to modify a number of nodes to one specific coordinate, for example Y=0, to ensure a true symmetry plane.

Method

Offset
allows you to select the coordinate direction and units of distance to offset the selected nodes.

Position
allows you to select the **Reference location** of the nodes to be moved and based on its position, modify the X, Y, and Z (or R, Theta, and Z) directions of the selected nodes. The reference location can either be an existing node, or a position on the screen.

Set
specifies the appropriate coordinate system and direction to move the nodes.

- Cartesian Coordinate System

  You can select the direction to move the nodes: X, Y, or Z and enter the number of units of distance.

- Cylindrical Coordinate System

  You can select the direction to move the nodes: R, θ, or Z and enter the number of units of distance.

Select Nodes
specifies the nodes selected. Select the nodes to be moved and click **Apply**. After selecting a node, its location coordinates will be displayed in the message window.

Offset Mesh

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elements
specifies the shell elements to be offset. Select the shell elements to offset in the normal direction. The normal direction of the elements, both the selected elements and the elements connected to those selected, must be consistent.

Distance
specifies the distance to offset the selected elements.

Note

Offset will change the location of nodes, but will not change the number (or definition) of selected elements. The new node location will be based on the normal direction of all the elements connected to this node.

Align Nodes

Reference Direction
defines the reference direction. Select two nodes to indicate the reference direction.

Nodes
specifies the nodes to be aligned by the specified reference direction.

Redistribute Prism Edge

The **Redistribute Prism Edge** option allows you to redistribute prism layers to a specified constant first layer thickness or a specified growth ratio spanning the local prism column thickness. This option works with prism layers as well as a combination of prism and hexa inflation layers. A prism layer is constrained by three parameters which calculate the fourth (see the Prism Meshing Parameters section). **Total Height** is set during prism growth, and can be varying if you left the **Initial Height** as zero. The number of layers can be set during the prism growth or adjusted with split prisms (see the Split Prisms option). An initial **Height Ratio** is set in the Prism Meshing Parameters, but can be adjusted by this **Redistribute Prism Edge** option. With **Total Height** and **Number of Layers** now fixed, you can use this option to adjust the **Initial Height** or **Height Ratio** and force the calculation of the other.

Note

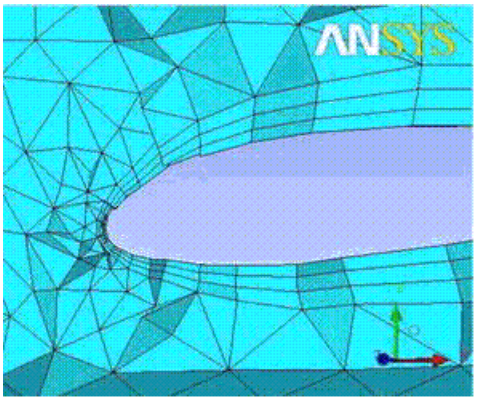
The redistribution will not move nodes connected to pyramids, but will redistribute other prisms in that layer, and above.

Method

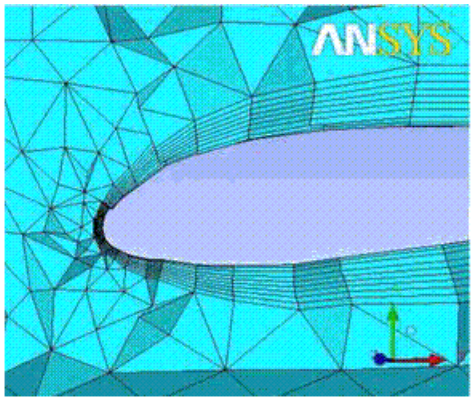
Fix ratio
allows you to specify a constant (fixed) growth ratio for redistributing the prisms. Note that the redistribution is calculated for each column of prisms; if the prism total height varies, the initial height will vary in order to maintain a fixed ratio.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

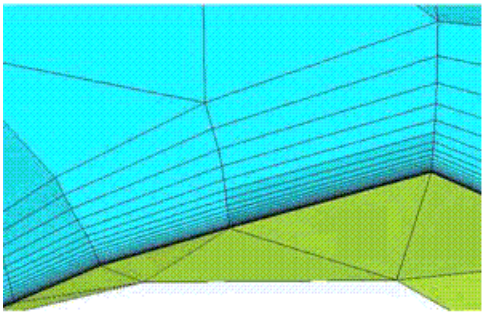
Figure 450: Prism Redistributed by Ratio



Initial Prism Mesh



Prism Mesh After Split



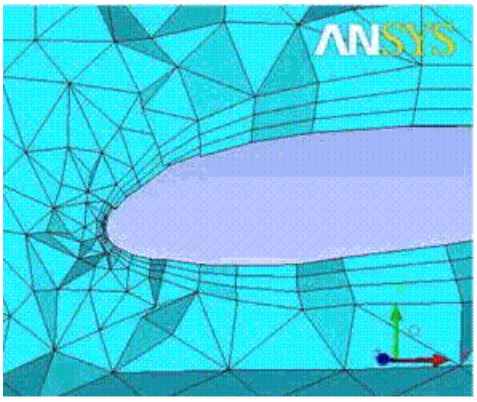
Prism Mesh Redistributed based on Ratio

Fix initial height

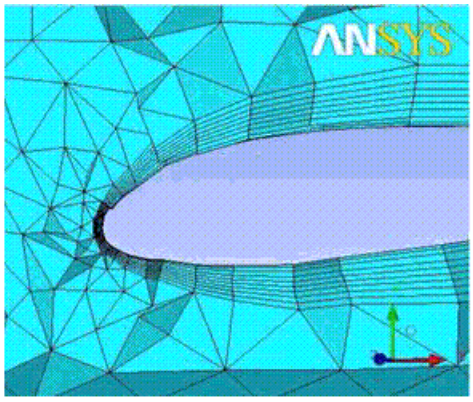
allows you to specify the desired initial height value of the prism layers in absolute units of the model, not the reference size. Note that the redistribution is calculated for each column of prisms, so specifying an initial height may result in varying ratios if the individual column **Total Height** varies.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

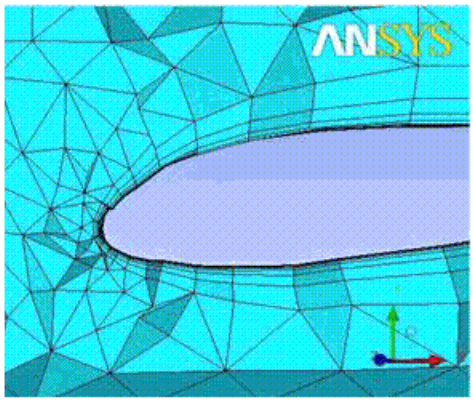
Figure 451: Prism Redistributed by Initial Height



Initial Prism Mesh



Prism Mesh After Split



Prism Mesh Redistributed based on Initial Height

Redistribute locked prism elements

when enabled, allows you to redistribute prism elements which have been locked (refer to Lock/Unlock Elements (p. 534) for details). When this option is disabled (default), the locked prism elements will be ignored during the redistribute operation.

Locked prism elements, if any, will be reported in the message window, when the **Redistribute Prism Edge** DEZ is opened. Locked elements can also be highlighted in the display by enabling **Mesh > Locked Elements** in the Display Tree.

Use local parameters

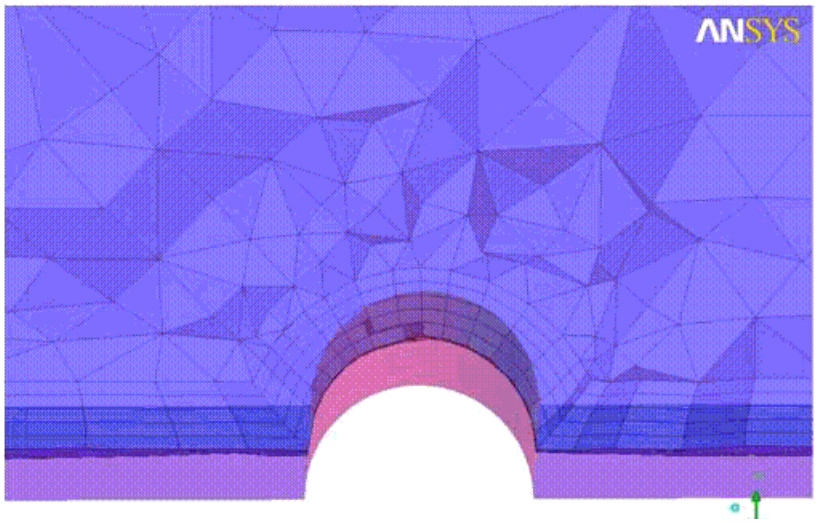
allows the use of the initial height or growth ratio defined locally on curves, surfaces, or parts for the prism mesh redistribution. If the **Use local parameters** option is enabled, and the local values are either not set or set to zero, the default value for the initial height or growth ratio will be used.

In Figure 452: Prism Redistributed with the **Use local parameters** Option (p. 531), the prism layer is redistributed based on initial height. The **Use local parameters** option is used, and the prism mesh redistribution accounts for the initial height set locally on the detail surface.

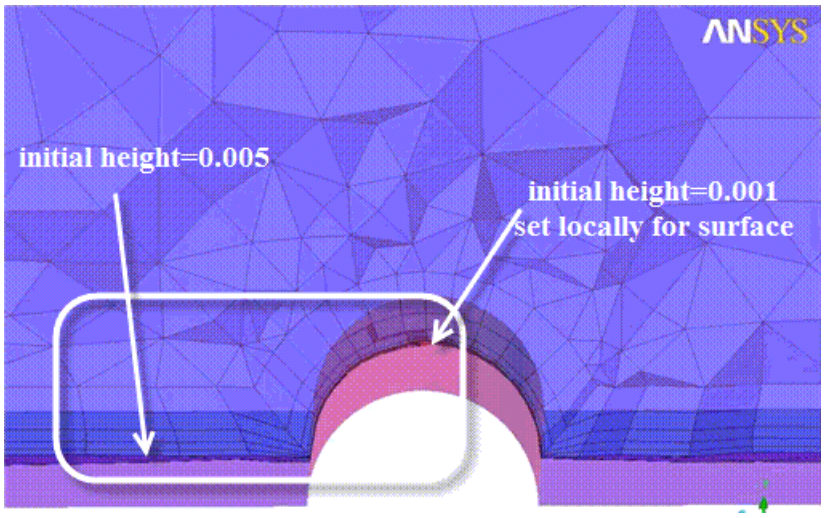
Figure 452: Prism Redistributed with the Use local parameters Option

Before Redistribution

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



After Redistribution



Project Node to Surface

The **Project Node to Surface** option allows you to project nodes to surfaces. Select from the following methods.

Nearest

Projects selected nodes to the nearest surfaces.

Direction

Projects selected nodes in a specified direction. If there is no surface in the specified direction, the node will be projected to the nearest surface along the specified vector.

Explicit

Specify the X, Y, and Z offsets to define the vector for the direction that the node is to be projected.

Vector

Select two points to define the vector for the direction that the node is to be projected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Always normal
If enabled, then the node will be projected only if the angle between the surface normal and the defined vector is less than 5 degrees. If the angle is greater than 5 degrees, the node will not be projected and a message will be displayed in the message window.

Project to
Specify whether to project nodes to **All Parts**, or to only **Active Parts**.

> **Note**
>
> Active parts are determined by the Parts branch of the tree. Active status is not affected by enabling and disabling entities or blanking specific entities. If you do not want nodes to be projected to certain entities, you may need to place those entities into a separate part and disable it (deactivate it) in the model tree.

Project Node to Curve

The **Project Node to Curve** option allows you to project the selected node to the nearest curve or a specified curve. The movement of the node will then be restricted along the curve.

Nodes to Project
Select the nodes to be projected.

Project by Tolerance
If enabled, all of the selected nodes that are closer to the selected curves by the specified tolerance are projected and moved to the curves. If disabled, the selected nodes will be projected and moved to the nearest curve.

Tolerance
Specify the tolerance value.

Curve(s)
Select the curves that the nodes are to be projected to.

Select
Select the nodes from the display.

Project Node to Point

The **Project Node to Point** option allows you to project a selected node to a particular point.

Nodes to Project
Select the nodes to project.

Projection Method

- **Explicit**

  To project nodes to a specified point.

  – **Point**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To project the node to a selected point.

– **Self**

To project the node to the point at the same location.

- **By Tolerance**

All of the selected nodes that are closer to the selected points by the specified tolerance are projected and moved to the points.

Un-Project Nodes

The **Un-Project Nodes** option allows you to un-project a projected node. The node will then be allowed unrestricted movement.

Select
Select the nodes from the display.

Lock/Unlock Elements

The **Lock/Unlock Elements** option allows you to lock or unlock elements. Locked elements will prevent automatic operations such as mesh smoothing or coarsening from moving the nodes. Locked elements can be highlighted by enabling **Mesh > Locked Elements** in the Display Tree.

Snap Project Nodes

The **Snap Project Nodes** option allows you to project nodes to their associated geometry. Select from the following methods.

Nearest
Projects selected nodes to the closest location on its associated geometry.

Direction
Projects selected nodes to their associated geometry along the specified direction.

Explicit
Specify the X, Y, and Z offsets to define the vector for the direction that the node is to be projected.

Vector
Select two points to define the vector for the direction that the node is to be projected.

Always normal
If enabled, then the node will be moved only if the angle between the surface normal and the defined vector is less than 5 degrees. If the angle is greater than 5 degrees, the node will not be projected and a message will be displayed in the message window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Project to
Specify whether to project nodes to **All Parts**, or to only **Active Parts**.

> **Note**
>
> Active parts are determined by the Parts branch of the tree. Active status is not affected by enabling and disabling entities or blanking specific entities. If you do not want nodes to be projected to certain entities, you may need to place those entities into a separate part and disable it (deactivate it) in the model tree.

Update Projection

The **Update Projection** option updates the projection of all nodes in the mesh. Nodes will be projected to a surface, curve or point.

Project Nodes to Plane

The **Project Nodes to Plane** option allows you to project a node to a plane.

> **Note**
>
> This option is similar to the **Project Nodes to Surface** option, except it does not require a surface and offers less control over how the projection is done.

Select
specifies the nodes selected for projection.

Plane Setup Method
specifies the method for defining the plane.

Point and Plane
defines the plane by a point and the normal. Select the point and specify the normal to the plane.

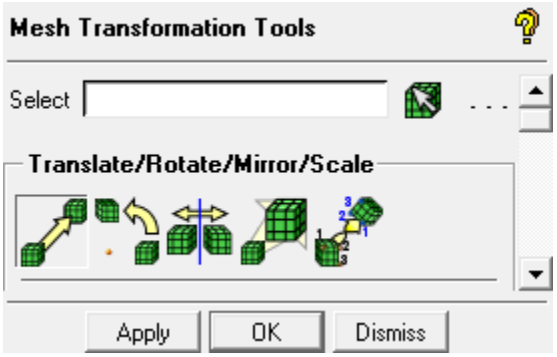
Three Points
defines a plane passing through the three specified points.

Transform Mesh

The **Transform Mesh** option can be used for transforming a portion or all of the mesh. This is useful when the mesh domain has symmetry. One section of the domain can be meshed and then rotated or translated in order to cover the complete domain.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 453: Mesh Transformation Tools Options



The different options are shown in Figure 453: Mesh Transformation Tools Options (p. 536). It is possible to use more than one method at a time.

Translate
Rotate
Mirror
Scale
Translate and Rotate

Translate

The **Translate** option is used to move elements laterally without rotation.

Select
specifies the elements for translation.

Copy
If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be generated at the selected location.

Number of copies
specifies the number of copies to generate. If at least one copy is generated, than you have the option to **Merge nodes**.

Increment Parts
To add the newly created copies of mesh to an existing part, click on the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes
automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for setting the **Tolerance Method** and **Deleting duplicate elements**.

Tolerance Method
allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete duplicate elements
deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Translation Method

- **Explicit**

  Enter the offset distance in the X, Y, and Z directions to translate the selected elements.

- **Vector**

  Define a vector by selecting two points. Elements will be moved in the direction of the defined vector.

Rotate

The **Rotate** option is used to rotate elements about a defined point. This is an extremely useful tool while meshing a circular domain. Mesh only one section of the domain, and rotate it to generate the complete domain.

Select
specifies the elements for rotation.

Copy
If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be rotated as specified.

Number of copies
specifies the number of copies to generate. If at least 1 copy is generated, than you have the option to **Merge nodes**.

Increment Parts
To add the newly created copies of mesh to an existing part, click on the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes
automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for setting the **Tolerance Method** and **Deleting duplicate elements**.

Tolerance Method
allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

Delete duplicate elements
deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Rotation Axis
specifies the axis of rotation. Select the axis of rotation as one of the coordinate axes, or a vector defined by two selected points.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angle
Enter the angle of rotation.

Center of Rotation
Define the point about which the elements will be rotated.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

- **Centroid**

  The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Mirror

The **Mirror** option relocates the mesh domain to its mirror image across a plane. A copy of the selected mesh can also be made as a mirror reflection while the original mesh is kept intact.

Select
specifies the elements to be mirrored.

Copy
If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be generated at the selected location. This will activate the option to **Merge nodes**.

Increment Parts
To add the newly created copies of mesh to an existing part, click on the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Merge nodes
automatically merges duplicated nodes when the mesh copies are adjacent to or overlapping one another. This uses a merge nodes tolerance setting to control the merge. Selecting merge nodes activates the options for the **Tolerance Method** and **Delete duplicate elements**.

Tolerance Method
allows you to determine how the merge tolerance should be specified. The **Automatic** option is recommended and will automatically calculate the tolerance as one tenth of the minimum edge length of all selected elements. If the **User defined** option is chosen, then an absolute tolerance can be set in the Tolerance field. The projection of the nodes are ignored during merging. There is also some internal code to prevent collapsing edges between nodes within the tolerance.

Delete duplicate elements
deletes duplicated elements if after merging nodes the copies overlap or are adjacent.

Reflection Plane Axis (Normal)
specifies the axis or vector that defines the normal to the mirror plane. The vector may pass through the plane.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Point of Reflection
Select the point about which the selected mesh elements are to be reflected.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

- **Centroid**

  The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Scale

The **Scale** option can be used to re-size the selected mesh domain by specifying factors in all three coordinate directions.

Copy
If enabled, the original mesh will be kept intact, and an exact copy of the mesh with duplicated nodes and elements will be resized as specified.

Increment Parts
To add the newly created copies of mesh to an existing part, click on the **Select Parts** icon. Select the part name from the window that opens up. If no part is selected, then the copies will be added to a new part.

Scale Mesh Factors
Enter the scale factor for the three directions X, Y, and Z.

Center of Transformation
Select the base point for scaling.

- **Origin**

  The origin of the model's global coordinate system, (0 0 0).

- **Centroid**

  The center of the bounding box of the selected elements.

- **Selected**

  Any point selected from the display.

Translate and Rotate

The **Translate and Rotate** option allows you to translate and rotate the mesh simultaneously. The reference location and target locations can be defined by 3 points, a curve or LCS and the mesh is translated and rotated to match.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy
If enabled, a copy of the selected entities will be created in the new location.

IncrementParts
To add the newly created copies of entities to a new part, click the **IncrementParts** icon. A window will open with the list of the parts of the selected entities. If a part is selected, the new entities will be placed in a new part named [old_part_name]_0. For multiple copies, each copy will be placed in a separate part, for example, MESH_1, MESH_2, MESH_3, etc.

Translate and Rotate Method
The three options for method are as follows:

3 points –> 3 points
Select six points in all. The first three points will be used as the reference for the entity to be transformed. The second set of three points is used to define the transformation. The result will match the first points of both sets, and the direction from the first to the second point, and the plane defined by the third point.

Curve –> Curve
Select two curves. The first curve is used as a reference for the entity to be transformed. The second curve is used to define the transformation. The result will match the beginning (parameter = 0) of both curves, the direction from parameter 0 to 0.5, and the plane defined by the end (parameter 1) of the curves. A curve used to define the transformation can be included in the entities selected to be transformed.

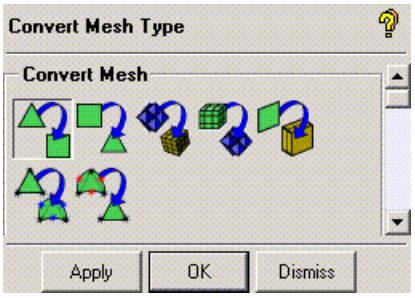
LCS –> LCS
Click **Show LCS** to view any defined coordinate systems. Select two local coordinate systems.

The result will match the origins and align the axes of the first LCS to the second LCS.

Convert Mesh Type

The **Convert Mesh Type** options are used to change one element type to another.

Figure 454: Convert Mesh Type Options



The available options are shown in Figure 454: Convert Mesh Type Options (p. 540).
Tri to Quad
Quad to Tri
Tetra to Hexa
All Types to Tetra
Shell to Solid
Create Mid Side Nodes
Delete Mid Side Nodes

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tri to Quad

The **Tri to Quad** option allows you to convert a triangular surface mesh to quadrilateral surface mesh. This is applicable for all visible surfaces.

Elements
specifies the triangular elements to be converted.

Surface Projection
contains options for projecting the newly created elements to the nearest entity of the geometry.

Quadrization
If this option is enabled, then the quadratic elements will be converted.

Note

Quadrization can only be done on the entire mesh.

Mesh Improvement
improves mesh quality after conversion. Select from the following range:

- 0 = No improvement steps.
- 1 = Stop improvement if less than 20% of the total elements are free triangles. This is the default value.
- 2 = Stop improvement if less than 2% of the total elements are free triangles.
- 3 = Perform as many improvement steps as needed to complete the conversion.

Max Skewness
specifies the maximum allowable skewness for conversion.

Max Warpage
specifies the maximum allowable warpage for conversion.

Quad to Tri

The **Quad to Tri** option divides each quad element into two triangles. The diagonal edge will be placed so as to make the minimum internal angle as large as possible.

Method

- **Shells Only**

  divides the quad elements only into triangle elements.

  – **Shells**

    specifies the shell elements to be converted.

- **Through Hexas**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

splits the attached hexa elements into two prisms and propagates the split into the volume. This will result in a column of hexas split into prisms.

– **Split Face Nodes**

splits the face nodes of quad element. Select diagonally opposite nodes of the quad element.

Tetra to Hexa

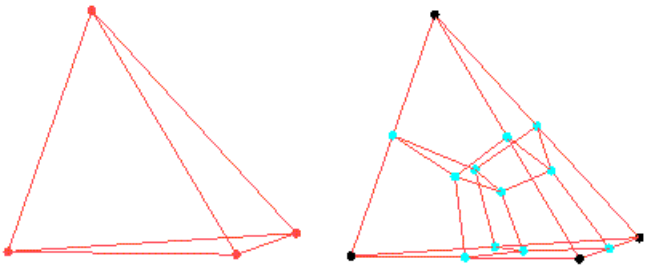
The **Tetra to Hexa** option allows you to convert tetrahedral mesh to hexahedral mesh.

Method

- **1 tetra to 4 hexa**

  This option will subdivide selected tetra elements into four hexa elements by creating a node at the center of the tetra, at the centroid of the tri faces of the tetra. The 4 hexa elements are created by connecting these nodes.

  Figure 455: Example of 1 Tetra to 4 Hexa



  Project nodes
  If disabled, the midface nodes that are created will be left linear and will not be projected. If enabled, the node will be projected in the manner determined by the **Normal to elements** option.

  Normal to elements
  If this is enabled, the node will be projected normal to the triangle face. If the angle between the normal direction and the projected point is greater than 5 degrees, then the node will be left unprojected. If this option is disabled, the node will be projected to the closest surface.

The **Project nodes** option will project new face nodes onto the geometry.

- **12 tetra to 1 hexa**

  Works on Octree generated mesh to convert groups of 12 tetras to 1 hexa element.

  Note

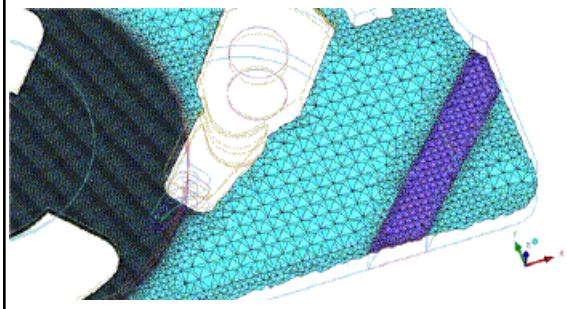
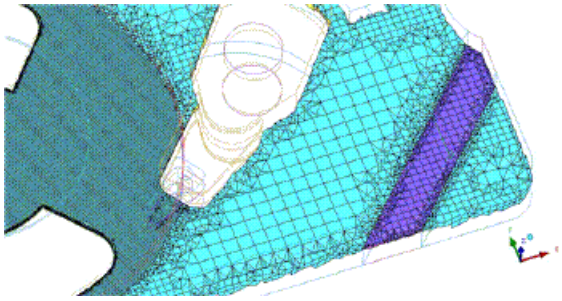
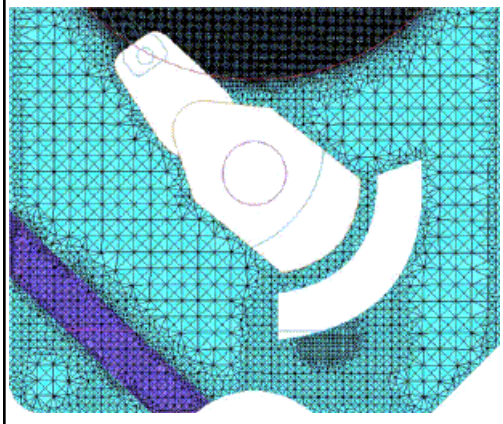
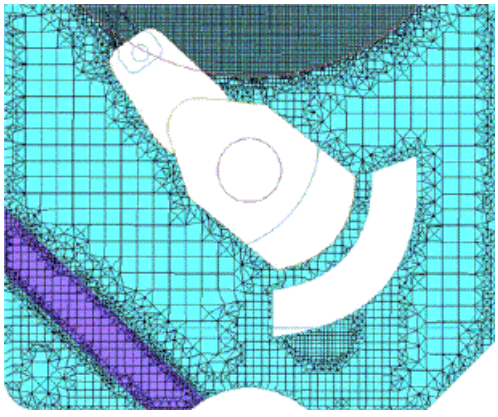
  This only works on Octree generated mesh because that algorithm started with Cartesian mesh which it converted to tetras with a 1 to 12 process.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The primary control is to specify the **Min aspect ratio** for the newly created hexa elements, clumps of tetras that would produce a hexa of lower quality are not converted. Pyramids are used to create a conformal interface between converted hexas and the surrounding tetra elements. Uniform regions of Octree mesh convert most effectively. Octree size transition regions cannot be converted; use density boxes to control where the size transitions happen (away from critical regions). Also, reducing the number of transitions (more uniform sizing) can result in a higher conversion rate. If the mesh against the surface is smoothed, it may not convert as well. You can disable surface smoothing (Octree tetra global mesh parameter) if you want hexas to the boundary. You can select either **All** tetra elements or elements from selected **Volume element parts** for the conversion.

In Figure 456: Example for Conversion of 12 Tetra to 1 Hexa (p. 543), the Octree tetra mesh for a disk drive (figure (A)) is converted to an Octree hexa hybrid mesh (figure (B)). This example has a large number of transitions as well as prism elements. These transitions, smoothing, and non Octree element types reduce the conversion rate. From the count of element types before and after the conversion (TETRA_4 : 4220472 converted to TETRA_4 : 2004355 and HEXA_8 : 179262), note that the final number of elements is reduced by approximately half without coarsening the mesh.

Figure 456: Example for Conversion of 12 Tetra to 1 Hexa

(A) Octree Tetra Mesh With Prisms for the Disk Drive Before Conversion	(B) Octree Hexa Hybrid Mesh for the Disk Drive After Conversion
	
	
Element types: NODE : 150 LINE_2 : 11493 TETRA_4 : 4220472 TRI_3 : 445817	Element types : NODE : 150 LINE_2 : 11726 TETRA_4 : 2004355 HEXA_8 : 179262

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PENTA_6 : 1788782 QUAD_4 : 2898 PYRA_5 : 11186	TRI_3 : 434319 PENTA_6 : 1748756 QUAD_4 : 11786 PYRA_5 : 176324

Note

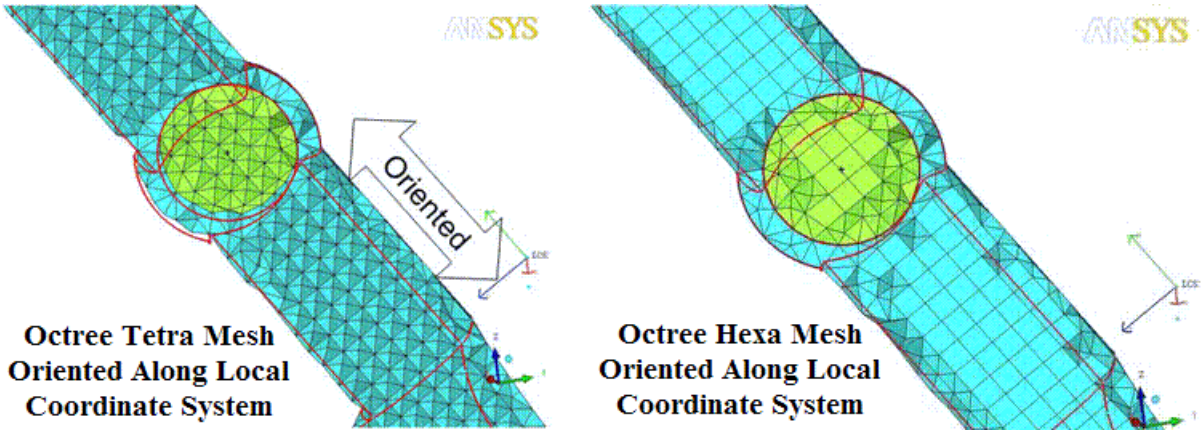
Only tetras are converted, not prisms. Also, prisms cannot grow into hexa elements, so you need to maintain a layer of tetras near walls where you intend to grow prisms or insert prism layers before using the 12 Tetra to 1 Hexa. conversion.

Aligning the Octree mesh with the principal model directions (such as along a pipe or wall) can further increase the conversion rate and result in hybrid mesh that is better aligned with the flow.

Note

The **Use active coordinate system** option must be enabled when converting oriented Octree tetra mesh to oriented hexa hybrid mesh.

Figure 457: LCS Oriented Octree Tetra Mesh Converted to LCS Oriented Hexa Mesh


Oriented
Octree Tetra Mesh Oriented Along Local Coordinate System
Octree Hexa Mesh Oriented Along Local Coordinate System

All Types to Tetra

The **All Types to Tetra** option converts all element types into tetra elements.

All to Tetra

Converts all elements into tetra elements. The selected elements do not have to be a specific initial element type.

Selected elements

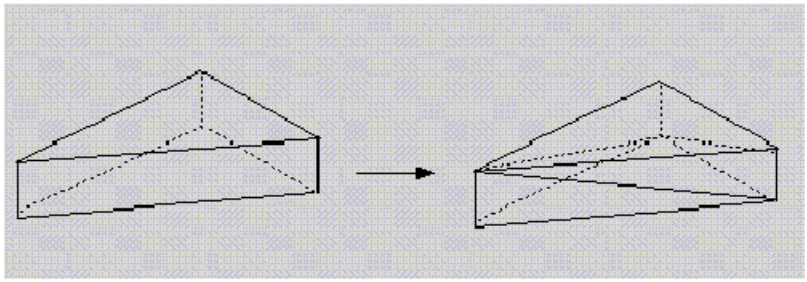
If enabled, you can select which mesh elements to convert to tetra elements. Otherwise the full mesh of all types of volume elements will be converted to tetra elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Make consistent
If enabled, then elements that are attached to mesh elements that are converted to tetra will be automatically edited to keep the mesh consistent.

1 Prism to 3 Tetra
Subdivides each prism element into three tetra elements whose edges extend along the quad faces of the former prism.

Figure 458: 1 Prism to 3 Tetra



Shell to Solid

The **Shell to Solid** option converts 2D elements into 3D elements.

Selected elements
If enabled, you can select which 2D elements are to be extruded into 3D elements.

Thickness
allows you to specify a thickness value of the 3D elements.

Square corners
If enabled, square corners will be formed. If elements at an L-bracked are being extruded, then by default the corner will be rounded, as the node extrusions are given an absolute valued based on thickness. This function will cause the node extrusions to be multipled by a factor of the angle, so that a sharp corner results. See Figure 459: Shell to Solid Conversion Without Sharp Corners Option (p. 546) and Figure 460: Shell to Solid Conversion With Sharp Corners Option (p. 546).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 459: Shell to Solid Conversion Without Sharp Corners Option

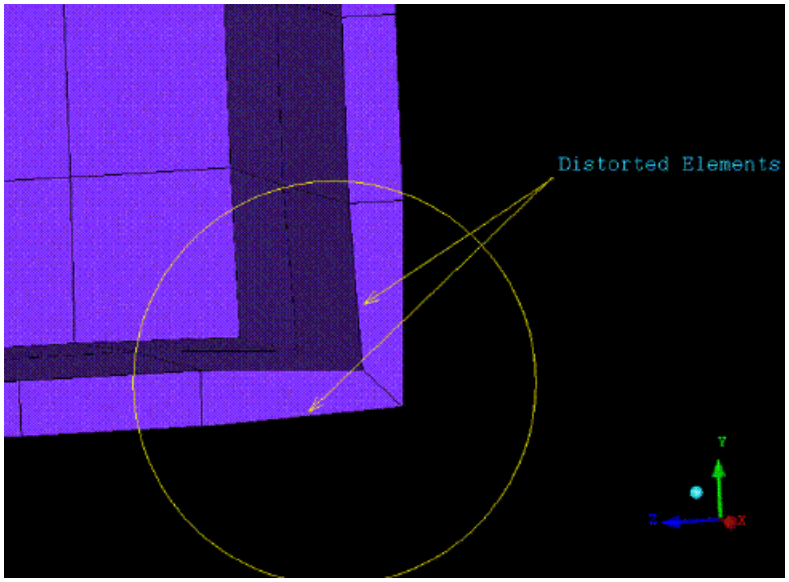
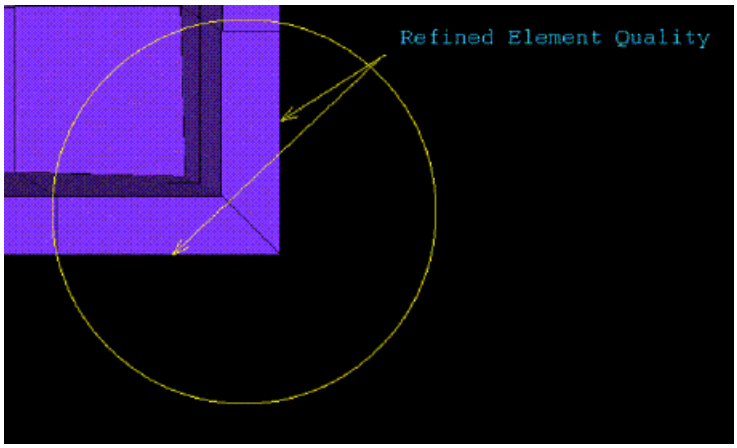


Figure 460: Shell to Solid Conversion With Sharp Corners Option



Hexa at T-connections

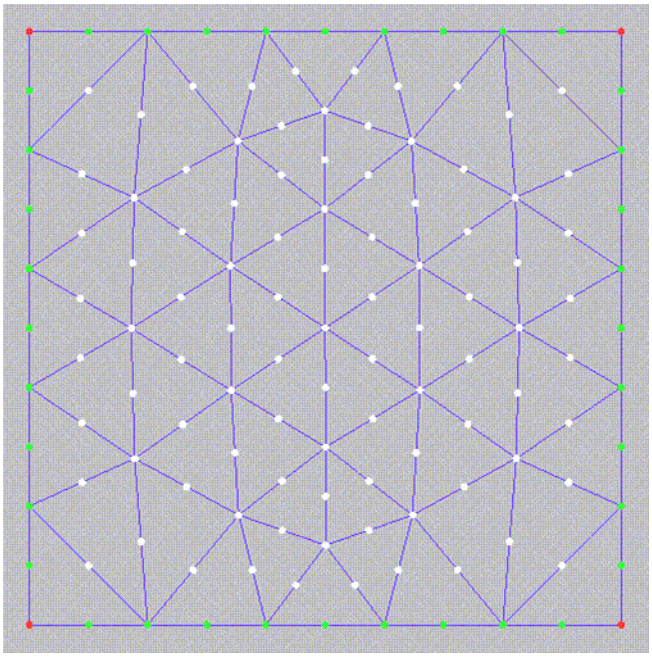
If enabled, hexa elements will be formed at T-connections.

Create Mid Side Nodes

The **Create Mid Side Nodes** option allows you to create mid side nodes on all the elements of the mesh, as shown in Figure 461: Create Mid Side Nodes Example (p. 547).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 461: Create Mid Side Nodes Example



Mid face node

If enabled, nodes will be created at the center of each face.

Create node on interface

This option is used when creating mid side nodes on the interface between selected elements and the elements attached to them. If enabled, then the edges of the attached elements will also become quadratic. If disabled, then the edges that are common to both the selected elements and attached elements will remain linear.

Project to geometry

If enabled, the newly created nodes will be projected to the nearest geometry.

Calculate projection

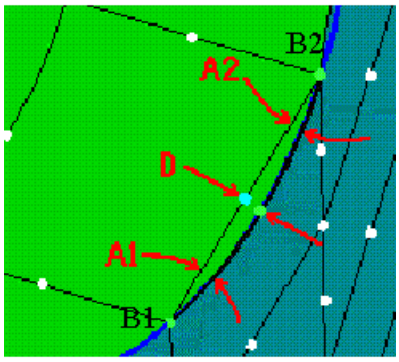
When a quadratic node is created, it will be projected to all marked curves or surfaces of its linear neighbors, and also the minimum projection will be calculated. The actual projection will be compared to the minimum projection and checked to see if it is within the specified tolerance. If not, the minimum projection will be used.

Check max

If any of the following three conditions is enabled, then the mid side node that exceeds the required condition(s) will not be projected. For the following max conditions refer to Figure 462: Check Max Conditions (p. 548).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 462: Check Max Conditions



- **Check deviation**

  The distance between the end nodes B1 and B2, multiplied with the value of the **Midnode max deviation** results in R. Next, imagine a line through mid node D, and perpendicular to line B1-B2. If the position of the projected node falls outside a distance R from the line, then the node will be moved on the geometry so that it falls within this range. If this is not possible, it will not be projected.

- **Check angle**

  This refers to the angles (A1 and A2) between the positions of the element edge before and after projection. If projecting a mid node would create an angle above the **Midnode max angle**, then the mid node will not be projected.

- **Check chord**

  The distance between the chord (B1-B2) and the projected mid node is restricted to the specified **Max deviation from chord**, defined in terms of % of the chord length. If this distance is exceeded then the node will be moved on the line (which is defined by the projected mid node and its projection to the chord) to the position of the maximum deviation.

Only selected elements

If enabled, you may select the elements to which the operation will be applied.

Tetra 10 elements

- **Standard check**

  If enabled, each projected mid node will be checked for negative values on the determinants at Gaussian integration points of all Tetra 10 elements to which the mid node belongs. If this check fails for at least one element then the dimension of the mid node will be increased, i.e., if it was projected to curves it will now be projected to surfaces, and then this check will be done again. Or if the mid node had been projected to surface, it will now be linearized.

- **Strong check**

  If enabled, a stronger check with respect to the Tetra 10 elements of the mid node will be done.

Automatic refinement

This is available only in the case of tetrahedral elements. For all elements a temporary mid node will be created on each element edge (with projection) and it will be checked with respect to some diagnostic

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

criteria for the attached tetrahedrals. If any of the criteria is not fulfilled, the element edge will be refined. In general it is not advisable to use this option.

Smooth refinement
If enabled, the mesh will be smoothed with 5 iterations to reach an aspect ratio of 0.25 for the tetra elements.

Delete Mid Side Nodes

The **Delete Mid Side Nodes** option allows you to delete mid side nodes. Mid Side Nodes can be deleted from the entire mesh or from selected elements/nodes in two different ways.

Elements or Nodes
Either elements or nodes can be selected.

Selected elements/nodes
If enabled, it allows you to delete mid side nodes from selected elements or to delete selected quadratic nodes.

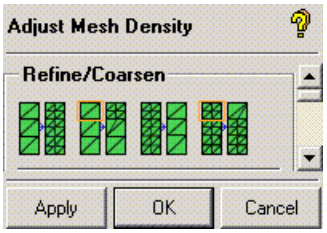
Convert Quadratic elements to multiple Linear elements
If disabled, this option will simply delete the midside/midface nodes without disturbing any other nodes. If enabled, this option will refine the mesh in such a way that all midside/midface nodes will become mesh nodes.

Keep interface node
This option is used when mid side nodes are deleted from selected elements that are attached to other quadratic elements. If enabled, then the selected elements will become linear, but nodes on the interface of the selected elements and the attached quadratic elements will remain.

Adjust Mesh Density

The **Adjust Mesh Density** options allow you to refine or coarsen the entire mesh or only at a particular location.

Figure 463: Adjust Mesh Density Options



The following options related to this feature are shown in Figure 463: Adjust Mesh Density Options (p. 549).
Refine All Mesh
Refine Selected Mesh
Coarsen All Mesh
Coarsen Selected Mesh

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Refinement
Refines, or subdivides, elements. This applies only to triangular surface elements, quads, and tetra elements. While refining triangular surface elements, if tetra elements are connected to the tri surface elements, the tetra's edges connected to the tri's will be subdivided so that the mesh remains conformal. Similarly, refining tetra elements will subdivide adjacent tetra or tri element edges so that the mesh remains conformal.

Coarsening
Coarsens a tetrahedral or tetra/prism hybrid mesh. Elements in the mesh will be coarsened by collapsing edges (merging nodes) and removing degenerate elements, but no operation will be performed that would yield an element with an aspect ratio worse than what is specified. The degree of coarsening (number of elements before vs. number of elements after) is automatically determined and reported in the Message window. Disconnected vertices are automatically deleted after coarsening.

Refine All Mesh

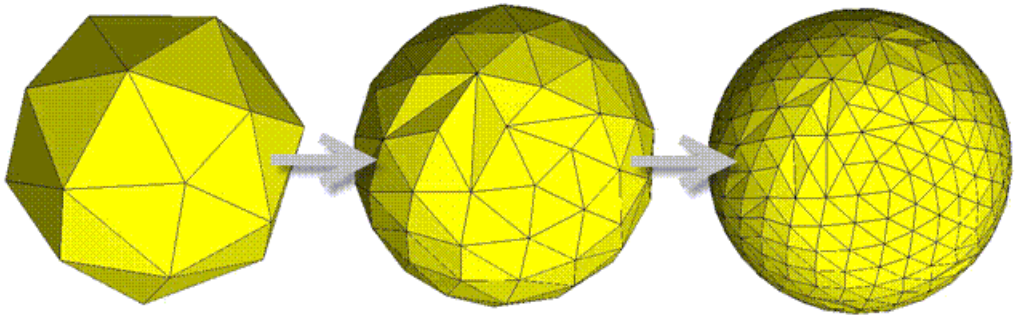
The **Refine All Mesh** option refines all of the visible displayed elements. The different methods are described below.

Pure refinement
Refines the mesh by splitting the edges and then performing swapping to improve element quality.

Note

In cases where adjacent tetra are connected by an edge only (not a face), the edge is split, but then cannot be swapped to improve the quality. This may result in lower element quality (see Figure 464: Refinement by Edge Splitting (p. 550)).

Figure 464: Refinement by Edge Splitting



Steps
specifies the number of refinement steps to be completed.

Project Nodes
projects the new nodes of the subdivided elements on the geometry.

Note

It is necessary to have a Tetin file loaded for the nodes to be projected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On Surface Only

allows you to refine only the surface mesh. This option only splits edges on the surface and then performs swapping to improve element quality.

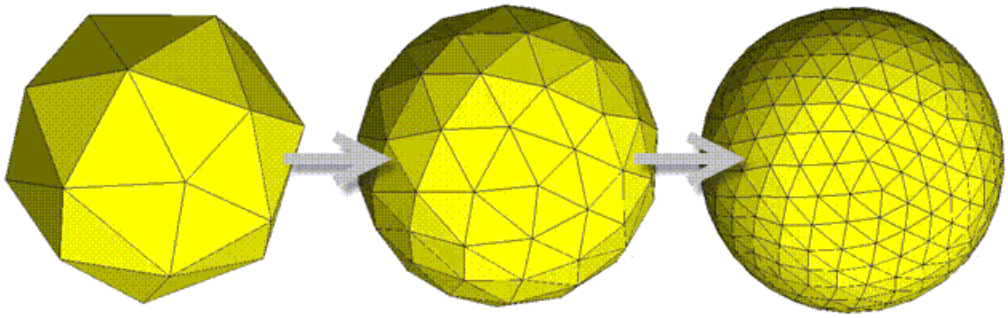
Tip

The **On Surface Only** option is useful when you need to refine only the surface mesh. As the volume mesh is not refined, the quality problems seen in Figure 464: Refinement by Edge Splitting (p. 550) will be avoided.

By Mid Side Nodes Only

refines the mesh using mid-side nodes. This option produces a more uniform mesh (see Figure 465: Refinement by Mid Side Nodes Only (p. 551)).

Figure 465: Refinement by Mid Side Nodes Only



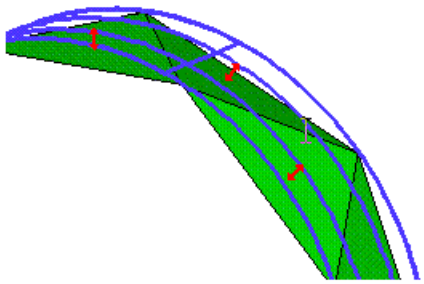
Note

The **By Mid Side Nodes Only** option can only be used globally, and not for local refinement.

Surface deviation

For surface elements (triangles), this option subdivides the elements based on the surface deviation computed. The surface deviation is the distance from the centroid of the element to the surface, in the direction normal to the surface as illustrated in Figure 466: Surface Deviation (p. 551).

Figure 466: Surface Deviation



Max Surface Deviation

specifies the maximum allowable surface deviation. If the computed surface deviation is greater than the defined value, the element will be subdivided.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Steps
is the number of refinement steps. The number of sub elements that are created is determined by the formula, 4^n, where **n** is the **Max steps** value. One step will break an element into 4 sub elements, 2 will yield 16 sub elements, etc.

On Surface Only
allows you to refine only the surface mesh.

Edge length

- **Max edge length**

  specifies the maximum edge length after refinement.

- **Weak refinement**

  If enabled, the refinement will not be propagated strongly into the neighboring regions.

- **Max steps**

  the number of refinement steps.

- **Project nodes**

  projects the new nodes of the subdivided elements on the geometry.

> **Note**
>
> It is necessary to have a Tetin file loaded for the nodes to be projected.

Refine Selected Mesh

The **Refine Selected Mesh** option refines selected elements from the mesh.

The methods for refining selected mesh are the same as described in Refine All Mesh (p. 550).

Coarsen All Mesh

The **Coarsen All Mesh** coarsens all of the visible displayed elements.

Min aspect ratio
the minimum aspect ratio (the circumsphere ratio) allowed for the resulting coarsened elements. The lower the minimum aspect ratio, the more elements in the mesh will be coarsened.

Max size
is the largest tetra size allowed by coarsening. Only elements below this size will be coarsened, and only up to this size. The larger this value, the more the mesh will be coarsened.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max surface deviation
For surface elements (triangles), this option coarsens the elements based on the distance from the centroid of the element to the surface, in the direction normal to the surface. If this distance is less than the defined **Max Surface Deviation**, the elements will be coarsened.

Number of Iterations
is the number of smoothing iterations. Smoothing will be automatically performed after the coarsening process if this number is nonzero.

Coarsen surface
If this is enabled, then surface triangular elements will also be coarsened. If disabled, only tetra and prism volume elements are coarsened.

Maintain surface sizes
If this is enabled, it will attempt to coarsen up to the sizes prescribed on the geometry (points, curves and surfaces). Local coarsening can be achieved this way by only changing tetra sizes on specific entities. When tetra mesh is generated, the elements are divided to these sizes. So if no tetra sizes are changed after generating the tetra mesh, then enabling this option will prevent any coarsening from being performed.

Parts to freeze
specifies the parts which will not be coarsened. Nodes of these parts will remain fixed. If there is only one volume part, and this part is frozen, no coarsening will be performed.

Coarsen Selected Mesh

The **Coarsen Selected Mesh** option coarsens selected elements from the mesh.

Note

This feature works for quad and quad dominant mesh, but not all tri mesh. In order to coarsen selected all tri or mostly tri mesh, freeze the parts that are not to be coarsened and then use **Coarsen All Mesh**.

Elements
specifies the elements selected for coarsening.

Min aspect ratio
the minimum aspect ratio (the circumsphere ratio) allowed for the resulting coarsened elements. The lower the minimum aspect ratio, the more elements in the mesh will be coarsened.

Max size
is the largest tetra size allowed by coarsening. Only elements below this size will be coarsened, and only up to this size. The larger this value, the more the mesh will be coarsened.

Max surface deviation
For surface elements (triangles), this option coarsens the elements based on the distance from the centroid of the element to the surface, in the direction normal to the surface. If this distance is less than the defined **Max Surface Deviation**, the elements will be coarsened.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

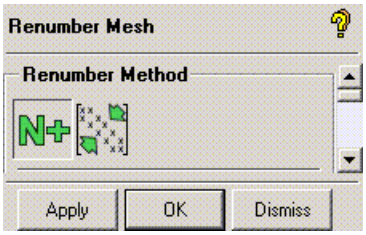
Number of Iterations
is the number of smoothing iterations. Smoothing will be automatically performed after the coarsening process if this number is nonzero.

Maintain surface sizes
If this is enabled, it will attempt to coarsen up to the sizes prescribed on the geometry (points, curves and surfaces). Local coarsening can be achieved this way by only changing tetra sizes on specific entities. When tetra mesh is generated, the elements are divided to these sizes. So if no tetra sizes are changed after generating the tetra mesh, then enabling this option will prevent any coarsening from being performed.

Renumber Mesh

The **Renumber Mesh** option renumbers all of the elements so that the cell numbering from minimum to maximum is along a defined direction. This is used to speed up the computation of the solution.

Figure 467: Renumber Mesh Options



The different options are shown in Figure 467: Renumber Mesh Options (p. 554).
User Defined
Optimize Bandwidth

User Defined

Renumber Elements
enables renumbering of the elements.

Starting element number
starts element numbering from the specified number.

Element number range
gives range of existing element numbers.

Renumber Nodes
enables the renumbering of the nodes.

Starting node number
starts node numbering from the given node number.

Node number range
gives range of existing node numbers.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

- **By Parts**

  renumbers elements and/or nodes one part at a time.

- **All elements/nodes**

  renumbers all the elements and nodes.

- **Selected Elements**

  renumbers only the selected elements.

- **Selected nodes**

  renumbers only the selected nodes.

Direction
the global axes direction in which to renumber elements or nodes. For the X axis, the direction is 1 0 0.

Skip 0-numbered Elements and Nodes
If enabled, the 0-numbered elements and nodes will be skipped for renumbering.

Optimize Bandwidth

The **Optimize Bandwidth** option renumbers nodes to minimize the bandwidth of the element/node matrix.

Iterations
number of iterations to reorder node numbers.

Profile
minimizes the number of elements in the profile of the global system matrix. This option supports solvers which eliminate operations on zeros outside the profile using a skyline storage scheme.

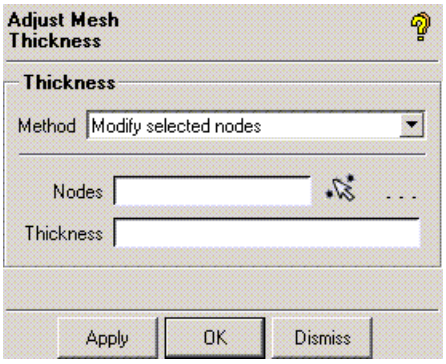
Bandwidth
minimizes the bandwidth of the global system matrix.

Assign Mesh Thickness

The **Assign Mesh Thickness** options available are shown in Figure 468: Adjust Mesh Thickness Options (p. 556)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 468: Adjust Mesh Thickness Options



Note

Uniform shell thickness can also be applied under **Properties > Define 2D Element properties**, but that definition is stored in an attribute file and not in the mesh. Thickness applied here becomes the default when setting up properties for 2D elements. Variable thickness should always be applied here before defining 2D element properties.

Method

- **Calculate**

  Mesh thickness will be assigned automatically to each node of the surface element from the thickness information stored on the surface geometry at that location. This thickness can come automatically from the midsurfacing process (including varying thickness) or be applied manually with **Repair Geometry > Adjust varying thickness**.

  The thickness is defined normal to the surface in both directions. The thickness can be viewed from the display tree by right clicking **Mesh > Shells > Shell Thickness**.

  Calculate from solid
  allows you to extract the thickness information of shell elements from the solid geometry around the shells. This is used when the midsurface thickness information is not present on the surfaces. To use this, load the original solid geometry. The shells mesh should be within this geometry. Select the part of the solid to calculate the thickness. This method uses the nearest point projection for thickness calculation. The average of the thickness in both directions is used for each node.

  Piercing
  is an alternative method for calculating the thickness from a solid. It may be preferable for surfaces with greater thickness where the nearest point method may not be appropriate.

  Note

  The nearest point projection and piercing methods lead to similar results for surfaces with small thickness.

- **Remove**

  Removes the mesh thickness assigned to the nodes of the surface elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

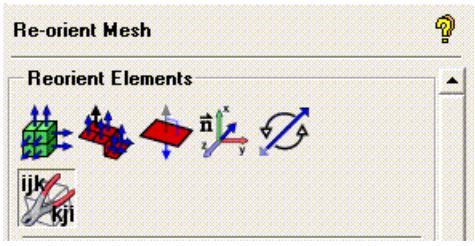
- **Modify selected nodes**

  Individual nodes of surface element can be assigned a specified thickness.

Reorient Mesh

The **Reorient Mesh** option changes the direction of the normals of selected elements or all the elements in a particular manner.

Figure 469: Reorient Mesh Options



The options for reorienting the mesh are shown in Figure 469: Reorient Mesh Options (p. 557).
Reorient Volume
Reorient Consistent
Reverse Direction
Reorient Direction
Reverse Line Element Direction
Change Element IJK

Reorient Volume

Reorients the normals of the displayed elements to point into the volumetric domain (inwards) or away from the domain (outwards). By default, all of the face normals will be reoriented to point into the domain, unless there are orientation errors.

Outwards

If toggled ON, all the normals will be oriented in the outward direction.

Reorient Consistent

Aligns all the normals of the displayed elements to have the same orientation (inwards or outwards) as the selected element. This works for all elements connected to the domain of the selected surface element. Any mesh disconnected from this mesh will not be affected. This operation will work across different parts.

Active parts
If toggled ON, only the elements in the parts that are turned ON in the Parts Display Tree will be reoriented.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverse Direction

Reverses the normals of the selected elements.

Elements
To select the elements from the display whose orientation is to be changed.

Filter by Screen Normal
This will reverse the normal of only those elements whose normal is in the opposite direction with respect to the screen view factor.

Reorient Direction

Changes the normal direction of the displayed elements according to a specified X Y Z vector. Since a face normal can only point in two possible opposing directions, the normal will point in the direction closest to that of the specified vector.

Reverse Line Element Direction

Reverses the line element direction.

Change Element IJK

There are four different methods available to change the IJK indices of an element.

IJK –> KIJ
For the selected element(s), the indices will be changed from IJK to KIJ.

Set Origin
For the selected element(s), the origin will be set at the specified node.

Align Element
Aligns all elements with reference to a selected element.

Set IJK
Allows you to set the current IJK indices of the selected element(s) to new IJK indices.

Set I
Select which of the current indices to mark as the new I index.

Set J
Select which of the current indices to mark as the new J index.

Set K
Select which of the current indices to mark as the new K index.

Delete Nodes

Deletes the selected visible nodes from the display.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete Elements

Deletes the selected elements from the display.

Edit Distributed Attribute

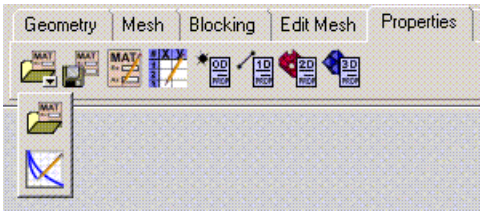
Makes the element with a distributed attribute visible. For example, if a model has distributed boundary conditions, this option allows you to view and edit the BC at an element or node basis.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties

Figure 470: Properties Menu



After a model is meshed, you may define or edit material and element properties using the options in the **Properties** menu. The options are as follows:

Load Material From File
Load Material From Library
Write Material File
Create Material Property
Create Material Property Table
Define Point Element Properties
Define 1D Element Properties
Define 2D Element Properties
Define 3D Element Properties

Note

In this section, the terminology used is the same as Nastran terminology. The Nastran manual may be a resource for terminology.

The following types of element properties can be defined. The elements are grouped by dimensionality.

- **0D (Point) element properties (MASS elements)**
- **1D (Line) element properties**
  - Bars
  - Rigid Beams
    - → RBAR
    - → RBE2
    - → RBE3
  - Rod Connection
  - Damper Connection

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Spring
  - Viscous Damper
  - Beam
  - Gap
  - Beam with Cross section
- **2D (Shell) element properties**
  - Shell
  - Shear
  - Layered Composite
- **3D (Solid) element properties**

Load Material From File

Existing material files can be loaded using the **Load Material From File** option, or using the **File > Parameters > Open Parameters** option. In either case, a file browser will open for you to select a material file (*.mat). Loaded material files will be listed in the **Display** tree, under **Material Properties**.

The following Nastran materials are supported:

- **MAT1** (Linear isotropic material properties)
- **MAT2** (Linear anisotropic material properties for two-dimensional elements)
- **MAT8** (Orthotropic material properties for isoparametric shell elements)
- **MAT9** (Anisotropic material properties for isoparametric solid elements)
- **MATS1** (Stress-dependent properties for nonlinear materials)
- **MATT1** (Temperature-dependent properties for MAT1 entry fields via TABLEMi entries)
- **MATT2** (Temperature-dependent properties for MAT2 entry fields via TABLEMj entries)
- **MATT9** (Temperature-dependent properties for MAT9 entry fields via TABLEMk entries)

Load Material From Library

The **Load Material From Library** icon is located on the pull-down menu under the **Load Material From File** icon. The following material files can be loaded from the Material Library:

- **Steel**
- **Magnesium Alloy**
- **Copper Alloy**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Aluminum Alloy**
- **Polyethylene**
- **Stainless Steel**
- **Concrete**
- **Titanium Alloy**
- **Grey Cast Iron**

The units for these materials are mm/C/MPa.

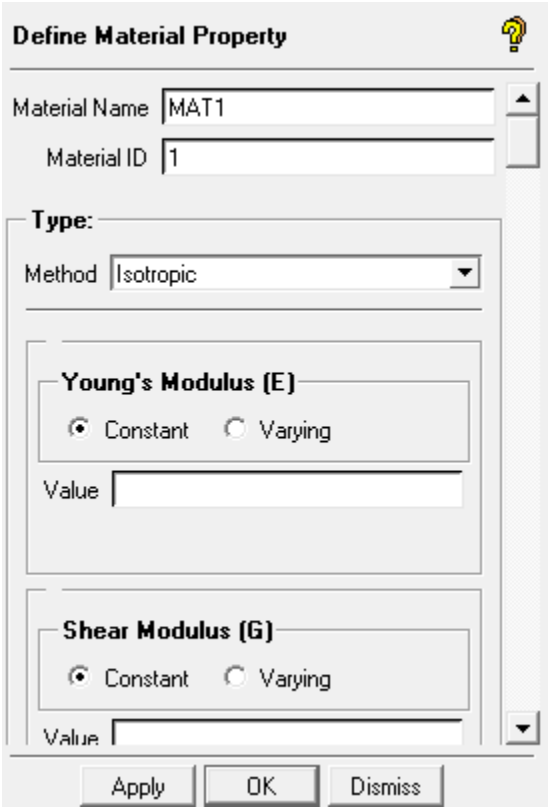
Write Material File

The **Write Material File** option allows you to save existing material property definitions to a material file (*.mat).

Create Material Property

The **Create Material Property** option opens a DEZ where you define the **Name**, **ID**, and **Type** of material.

Figure 471: Define Material Property DEZ



Material Name

specifies the name of the material being defined.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material ID
specifies the ID of the material (MID). Nastran property cards reference this ID.

Type
Each type has its own material properties DEZ, and is described in the subsequent sections.

The different material types are:
Isotropic
Shell Element Anisotropic
Solid Element Anisotropic
Shell Element Orthotropic
Isotropic Thermal Material
Anisotropic Thermal Material

Isotropic

Each material property described below can be selected as **Constant** or **Varying**. For varying properties, a material property table can be defined separately as described in Create Material Property Table (p. 566).

Young's Modulus (E)
Within the elastic limit, the ratio of direct stress to the strain produced is called the Young's modulus (E).

Shear Modulus (G)
Within the elastic limit, the ratio of shear stress to shear strain.

Typically, this field is left blank. Nastran will calculate the value of Shear Modulus internally, based on the following formula:

E = 2*(1+NU)*G,

where NU is the value of Poisson's ratio.

Poisson's Ratio (NU)
The ratio of lateral strain to longitudinal strain.

Mass Density (RHO)
The density of a material is its weight per unit volume (in SI units). This value will be used to automatically calculate the mass of all structural elements.

This value should be consistent with PARAM, WTMASS card value for Nastran runs.

Thermal Expansion Coefficient (A)
This coefficient (A) is used to calculate thermal strains when thermal loads exist on the structure.

Reference Temperature (TREF)
The reference temperature for the calculation of thermal loads or a temperature dependent thermal expansion coefficient. Typically, it is defined at room temperature in degrees Kelvin.

Structural Element Damping Coefficient (GE)
This value is found by multiplying the critical damping ratio C/C_0 by 2.0.

Stress Limits for Tension (ST)
This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stress Limits for Compression (SC)
This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

Stress Limits for Shear (SS)
This is an optional value, used only to compute margins of safety in certain elements and has no effect on the computational procedures.

Material Coordinate System (MCSID)
This is used only for PARAM, CURV processing.

LS Dyna Material Type
This specifies the LS Dyna Materials type.

Shell Element Anisotropic

Anisotropic materials have different material properties in the horizontal and vertical directions.

The properties of anisotropic shell elements include the same material properties that are described in the Isotropic (p. 564) Material Property section. Additional properties are described below:

G_{ij}
Elements from the material property matrix for this material type as described in the Nastran manual.

Thermal Expansion Coefficient Vectors (A_i)
These values are the thermal expansion coefficient vectors.

Solid Element Anisotropic

The properties of anisotropic solid elements include the same material properties described in the previous two sections. The properties that are unique to this material type are described below:

G_{ij}
Elements of the 6 x 6 symmetric material property matrix for this material type in the Nastran material coordinate system.

Shell Element Orthotropic

Orthotropic materials have three mutually perpendicular planes of elastic symmetry.

Modulus of Elasticity – Longitudinal (E1)
Modulus of elasticity in the longitudinal direction. It also defined as the fiber direction or 1-direction.

Modulus of Elasticity – Lateral (E2)
Modulus of elasticity in lateral direction. It also defined as the matrix direction or 2-direction.

Poisson's Ratio (NU12)
The ratio of lateral strain to longitudinal strain.

G12
In-plane shear modulus.

G1Z
Transverse shear modulus in the 1-Z plane.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

G2Z
Transverse shear modulus for shear in the 2-Z plane.

Mass Density (RHO)
The density of a material is its weight per unit volume (in SI units).

Thermal Expansion Coefficient (A_i)
Thermal expansion coefficient in the i direction. (Real)

Reference Temperature (TREF)
The reference temperature for the calculation of thermal loads or a temperature dependent thermal expansion coefficient.

Xt and Xc
Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. These values are required if a failure index is desired.

Yt and Yc
Allowable stresses or strains in tension and compression, respectively, in the lateral direction. These values are required if a failure index is desired.

S
Allowable stress or strain for in-plane shear.

Structural Damping Coefficient (GE)
This value is found by multiplying the critical damping ratio C/C_0 by 2.0.

F12
Interaction term in the tensor polynomial theory of Tsai-Wu. Required if the failure index by the Tsai-Wu theory is desired and if the value of F12 is different from 0.0.

STRN
For the maximum strain theory only. It indicates whether Xt, Xc, Yt, Yc and S are stress or strain allowable. (Real = 1.0, strain allowable; blank for stress allowable)

Isotropic Thermal Material

Isotropic Thermal Material corresponds to MAT4 (Heat Transfer Material properties, Isotropic) in Nastran. Using this option allows you to set one thermal conductivity for the model.

Anisotropic Thermal Material

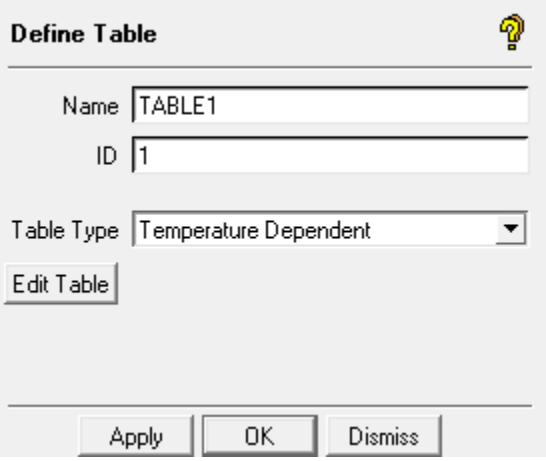
Anisotropic Thermal Material corresponds to MAT5 (Thermal Material property definition) in Nastran. Using this option allows you to set thermal conductivity for each plane.

Create Material Property Table

The **Create Material Property Table** option allows you to create a material property table for defining varying material properties.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

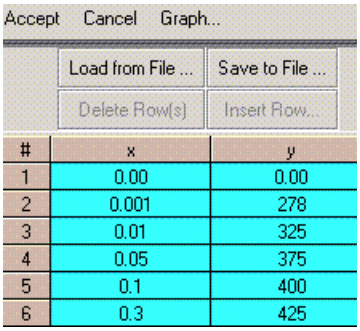
Figure 472: Define Material Property Table



Varying material properties can be defined with one of the following types of tables:

Enter the name and ID of the table, and select the table type. Click **Edit Table** to view the Table Editor, as shown below.

Figure 473: Material Properties Table Editor

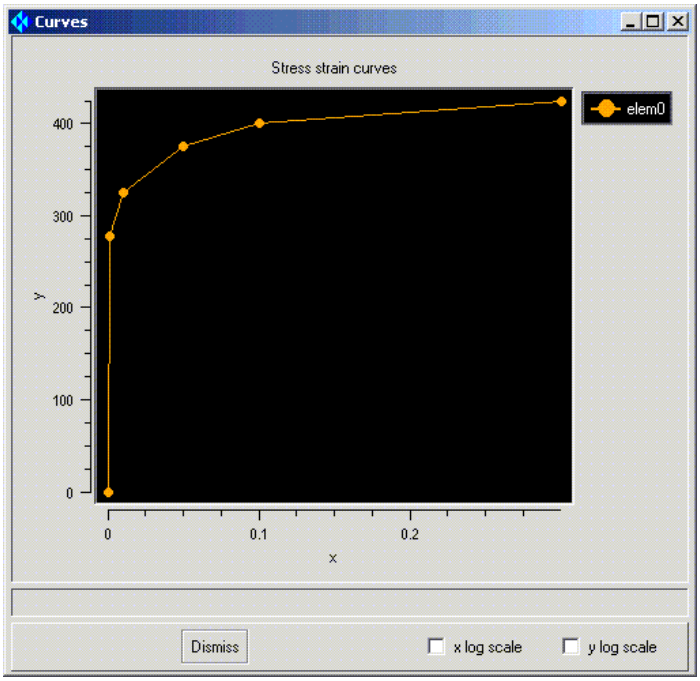


The following options are available in the **Table Editor** window:

- Define and save material properties by filling in the table entries.
- Load material properties from a file in a space delimited x-y data format.
- Display the material properties graph as shown in Figure 474: Material Properties Curve (p. 568)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 474: Material Properties Curve

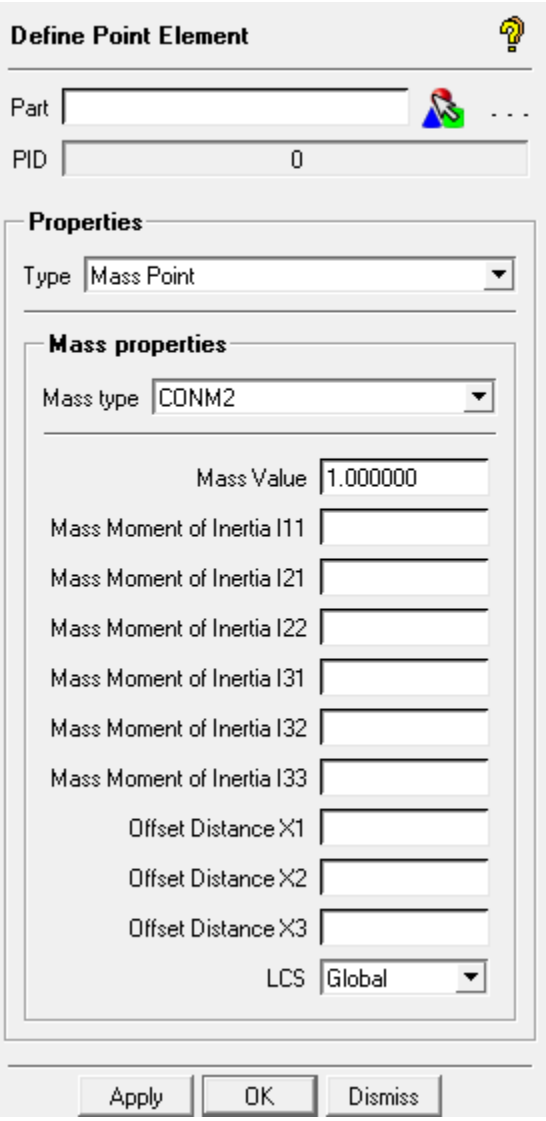


Define Point Element Properties

The **Define Point Element Properties** option allows you to define point element properties. These properties correspond to Nastran CONM2 element properties, which are mainly to define the mass values.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 475: Define Point Element DEZ



By defining individual point masses as separate parts, they are conveniently grouped under the **Display** Tree for easy manipulation.

Part
specifies the part for which the point properties will be assigned.

PID
This is grayed out by default.

Properties Type
The only type of point element property is **Mass Point**.

Mass Type
There are two options for Mass Types:

- **CONM1**

  The general form of defining a mass element.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CONM2**

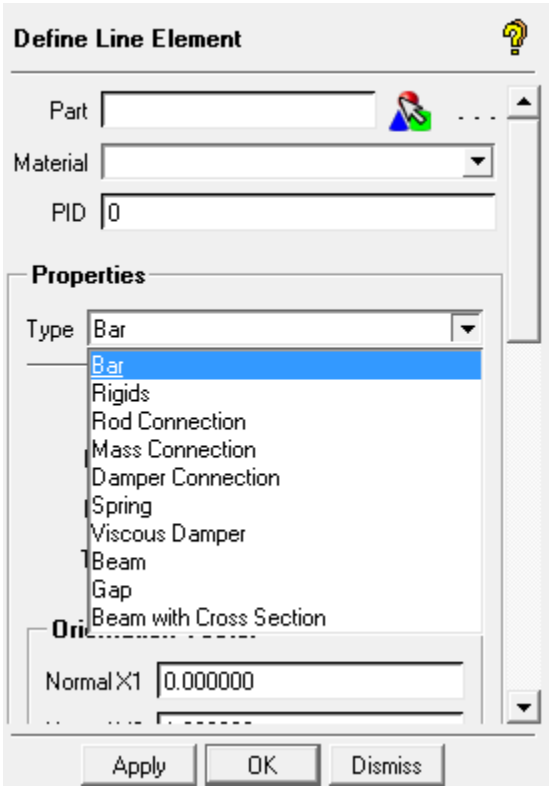
  A more specific means of defining a mass element.

Scalar Mass
specifies the value for the scalar mass (SI units).

Define 1D Element Properties

The **Define 1D Element Properties** option allows you to define line (curve) element properties. Choose the Type of 1D element from the drop-down list. Each type has its own DEZ.

Figure 476: Define Line Element DEZ



The following sections describe the types of 1D (line) element that can be defined.

Bar Element Properties
Rigid Elements Properties
Rod Connection Properties
Mass Connection Properties
Damper Connection Properties
Spring Properties
Viscous Damper Element Properties
Beam Element Properties
Gap Element Properties
Beam With Cross Section

Bar Element Properties

The properties for elastic bar elements are as follows:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The X axis of the LCS follows the bar from end A to end B. The Y axis of the LCS is used to calculate I2 (Iyy) and the Z axis for I1 (Izz).

The formulas that are referenced are available in Mechanics of Materials books or can be computed by using section analysis programs.

Part
specifies the part name to which the defined property will be assigned.

Material
specifies the material name for the defined property.

PID
specifies the property ID. Nastran property cards reference this ID.

Type
Bar will be automatically selected.

Cross Section Area
the value of the cross sectional area of the geometry.

Moment of Inertia (I1)
the moment of inertia for the Z axis (Izz).

Moment of Inertia (I2)
the moment of inertia for the Y axis (Iyy).

Torsional Constant (J)
the torsional constant for the geometry. For a circular cross section, this value is the sum of Izz and Iyy.

Normal X1, X2, X3
are the components of the orienting vector.

LS-DYNA Beam formulation
Choose the LS-DYNA beam type.

Rigid Elements Properties

The three types of rigid elements are as follows:

Rigid Bar (RBAR)
Rigid Bar with Single Dependent Nodes (Nastran CBAR elements with associated PBAR cards).

Dependent DOF (CMA)
specifies the required degree of freedom for one end of the bar.

Dependent DOF (CMB)
specifies the required degree of freedom for the other end of the bar.

Independent DOF (CNA)
specifies the required Independent degree of freedom for one end of the bar.

Independent DOF (CNB)
specifies the required Independent degree of freedom for the other end of the bar.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the required degree of freedom for one end of the bar.

Rigid Body (RBE2)
Rigid Body with Multiple Dependent Nodes (Nastran RBE2 elements).

Dependent DOF
specifies the required degree of freedom for one end of the bar.

Rigid Body (RBE3)
Rigid Body with Multiple Independent Nodes having weighted average of motion (Nastran RBE3 elements).

Dependent DOF
specifies the dependent degree of freedom.

Independent DOF
specifies the independent degree of freedom.

Weighting Factor
specifies the weighting factor. The default is 1.

Rod Connection Properties

The properties for elastic rod connections are as follows:

Cross Sectional Area
the cross sectional area of the geometry.

Torsion Constant
the torsion constant of the element.

Torsion Coefficient
the coefficient that determines torsional stress.

Nonstructural Mass/Unit Length
nonstructural mass per unit length.

Mass Connection Properties

The properties for mass connections are as follows:

Scalar Mass
the value of scalar mass of the element.

Component C1 and C2
the component numbers.

Damper Connection Properties

This corresponds to the Nastran property card PDAMP.

Force/Unit Velocity
is the damping factor to be supplied.

Components C1 and C2
damping components.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LS-DYNA Material Type
Select either Type 66 or Type 74.

Spring Properties

A spring element may be defined by the properties of **Stiffness**, **Damping Coefficient** and **Stress Coefficient**. This is associated with the Nastran PELAS card, which is used in conjunction with the CELAS1 and CELAS3 element connectivity cards.

Select LS-DYNA Material as either Type 66 or Type 74.

Note

It is customary to define spring elements for each degree of freedom separately. For a six degree of freedom bushing, you need to define 6 CELAS1 and 6 corresponding PELAS cards. Also, spring elements should be defined by coincident nodes (zero length) so that they don't transfer unintended moments.

Viscous Damper Element Properties

Viscous damping element properties define the PVISC card needed for CVISC viscous damper elements.

Damping for Extension
the damping factor for extension.

Damping for Rotation
the damping factor for rotation.

LS-DYNA Material Type
Select either Type 66 or Type 74.

Beam Element Properties

The properties of a beam element are described below. This element may be used to model tapered beams.

Cross Section Area
cross sectional area of the beam.

Moment of Inertia (I1)
area moment of inertia for bending in plane 1 about the neutral axis.

Moment of Inertia (I2)
area moment of inertia for bending in plane 2 about the neutral axis.

Moment of Inertia (I12)
This value should be 0.0

Torsional Constant (J)
is the torsional stiffness parameter.

Normal X1
the normal value for the X direction.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normal X2
the normal value for the Y direction.

Normal X3
the normal value for the Z direction.

LS-DYNA Beam formulation
specifies the LS-DYNA Beam formulation.

Gap Element Properties

The gap element properties are as follows:

Initial Gap (U0)
the initial gap opening.

Preload (F0)
is the preload.

Axial Stiffness of Closed Gap (KA)
axial stiffness for the closed gap.

Axial Stiffness of Open Gap (KB)
axial stiffness for the open gap.

Transverse Stiffness (KT)
transverse stiffness when the gap is closed. It is recommended that KT $\geq$ (0.1 * KA).

Coeff of Static Friction (MU1)
coefficient of static friction for the adaptive element or coefficient of friction in the y transverse direction for the non adaptive gap element.

Coeff of Kinetic Friction (MU2)
coefficient of kinetic friction for the adaptive gap element or coefficient of friction in the z transverse direction for the non adaptive gap element.

Max Allowable Penetration (TMAX)
maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment.

Max Allowable Adjustment Ratio (MAR)
maximum allowable adjustment ratio for adaptive penalty values KA and KT.

Fraction of TMAX Defining Lower Bound (TRMIN)
fraction of TMAX defining the lower bound for the allowable penetration. (Default = 0.001)

Normal X1
the normal value for the X direction.

Normal X2
the normal value for the Y direction.

Normal X3
the normal value for the Z direction.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beam With Cross Section

The Beam with Cross Section element properties are as follows:

Cross Section Group (GROUP)
specifies the Group Name.

Cross Section Type (TYPE)
the various types of beams that can be used are **I, T, L, H, BOX, BAR, ROD, TUBE, HAT, CHAN**.

Nonstructural Mass Per Unit Length (NSM)
Specify this value.

Set Cross Section Dimensions
specifies the cross section dimension values in the table.

> **Note**
>
> Every cross section type beam has its own **Set Cross Section** table.

Normal X1
the normal value for the X direction.

Normal X2
the normal value for the Y direction.

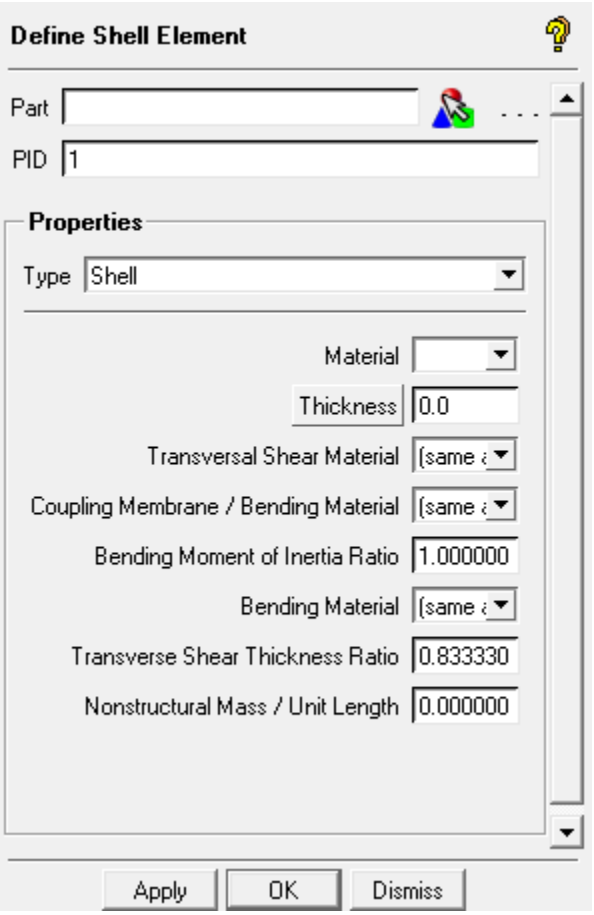
Normal X3
the normal value for the Z direction.

Define 2D Element Properties

The **Define 2D Element Properties** option allows you to define shell (surface) element properties. Choose the Type of 2D element from the drop-down list. Each type has its own DEZ.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 477: Define Shell Element DEZ



The following sections describe the 2D (Shell) Elements that can be defined:

Shell Elements
Shear Elements
Layered Composite Elements

Select the part to which the defined property will be assigned, and the PID of the property. Nastran property cards reference this ID.

Shell Elements

The properties for thin shells possessing membrane, bending, and transverse shear and coupling properties (Nastran PSHELL cards) are as follows:

Material
specifies the material.

Thickness
specifies the thickness of the geometry.

Transversal Shear Material
By default, this is the same material selected for the **Material** option. You can however select a different material.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coupling Membrane/Bending Material
By default, this is the same material selected for the **Material** option. You can however select a different material.

Bending Moment of Inertia Ratio
bending moment of inertia of the element.

Bending Material
By default, this is the same material selected for the **Material** option. You can however select a different material.

Transverse Shear Thickness Ratio
is the ratio of transverse shear thickness.

Nonstructural Mass/Unit Length
is the nonstructural mass per unit length.

Shear Elements

The properties for shear elements (Nastran PSHEAR cards for CSHEAR element types) are as follows:

Material
specifies the material.

Thickness
specifies the thickness of the geometry.

Nonstructural Mass/Unit Length
is the nonstructural mass per unit length.

Layered Composite Elements

The properties of layered composite elements (Nastran PCOMP card) are as follows:

Distance reference plane to bottom Surface (Z0)
is the distance from the reference plane to the bottom surface.

Nonstructural Mass Per Unit Area (NSM)
is the nonstructural mass per unit area.

Allowable Shear Stress in Bonding (SB)
allowable interlaminar shear stress.

Failure Theory (FT)
The following Failure Theory options are available: **HILL**, **HOFF**, **TSAI**, **STRN**.

Reference Temperature (TREF)
is the reference temperature.

Damping Coefficient (GE)
the damping coefficient.

Laminate Options (LAM)
The following options are available:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

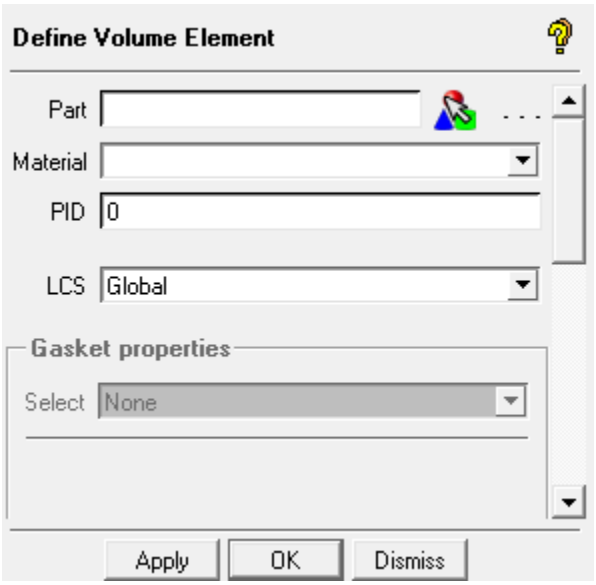
- **Blank**
- **SYM**
- **MEM**
- **BEND**

Number of Layers
specifies the number of composite layers.

Define 3D Element Properties

For 3D solids like HEXA, PENTA, and TETRA shaped elements, no special property is required except for the identification of the Part name, Material, PID, and the Local Coordinate System (LCS).

Figure 478: Define Volume Elements DEZ

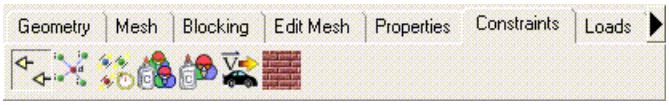


Specify the coordinate system on which this property should have applied.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constraints

Figure 479: Constraints Menu



The **Constraints** menu contains options for defining constraints. The constraints may be defined either on the geometry, or on the finite element model, or both. The methods to define constraints on the model are described in the following sections:

Create Constraint / Displacement
Create Constraint Equation
Define Constrained Node Sets
Define Contact
Define Single Surface Contact
Define Initial Velocity
Define Planar Rigid Wall

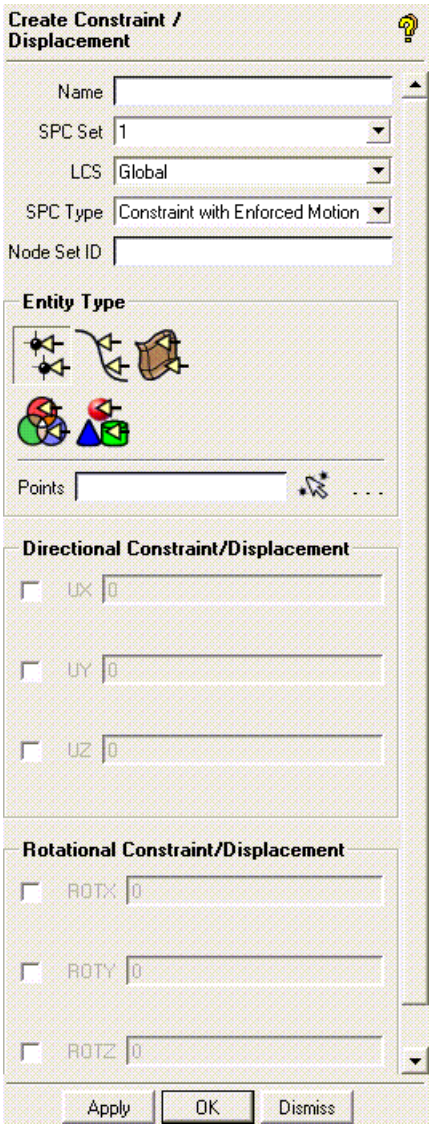
Create Constraint / Displacement

Constraints or displacements can be placed on different entity types in the geometry.

As soon as the constraint is created, it will be added to the Display Tree under **Constraints**. Constraints are grouped by sets. Multiple sets can be created with different constraints.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 480: Create Constraint / Displacement Window



Name
The constraint name.

SPC Set
is the number given to the constraint set. This is common Nastran Terminology.

LCS
the coordinate system for the applied constraint or displacement.

SPC Type
specifies the SPC Type.

Node Set ID
specifies the ID number of the Node Set.

Entity Type
the type of entity to which the constraint or displacement will be applied.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Directional Displacement

You can restrict the movement of the selected entities by disabling the fields. When enabled, you can specify the specific value of the displacement.

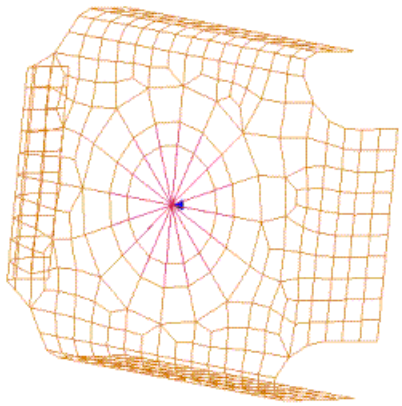
Rotational Displacement

You can restrict the movement of the selected entities by disabling the fields. When enabled, you can specify the specific value of the displacement.

Create Constraint / Displacement on Point

In the example in Figure 481: Constraints on Points or Nodes (p. 581), a constraint is placed on the center node of the model.

Figure 481: Constraints on Points or Nodes

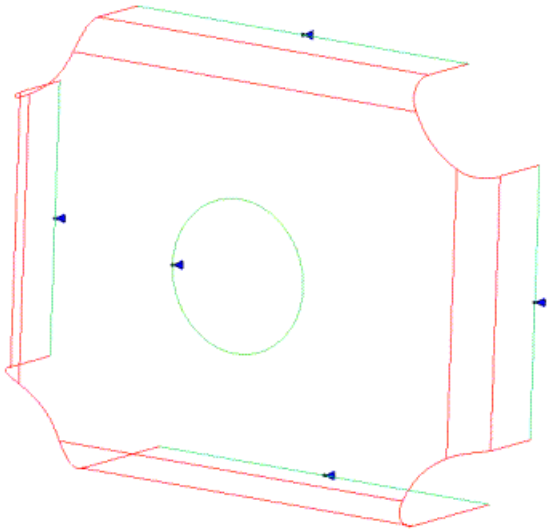


Create Constraint / Displacement on Curve

Constraints can also be applied on lines or edges of the parts to restrict its movement in desired degrees of freedom, as shown in Figure 482: Constraints on Curves (p. 582).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

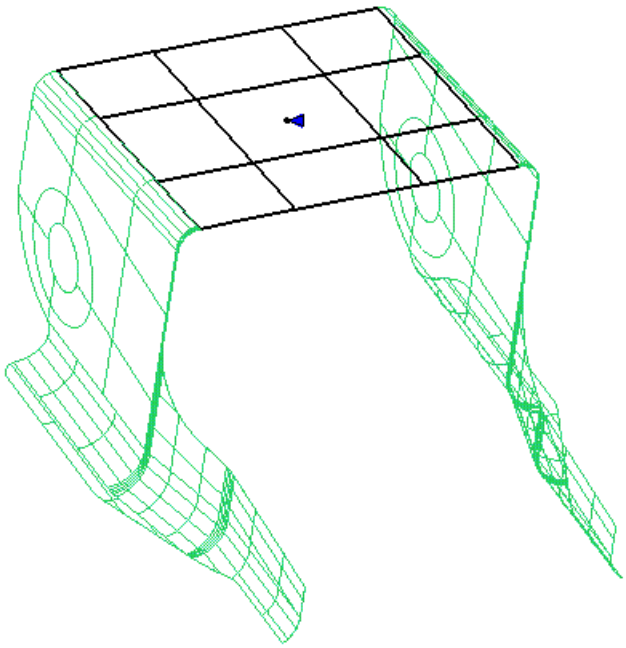
Figure 482: Constraints on Curves



Create Constraint / Displacement on Surface

Constraints can also be placed on surfaces. In Figure 483: Constraints on Surfaces (p. 582), the constraint is placed on the highlighted surface.

Figure 483: Constraints on Surfaces



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Constraint / Displacement on Subset

Constraints can also be placed on subsets of parts. Select the subsets on which the constraint should be applied.

Note

The subset must already be created.

Create Constraint / Displacement on Part

Constraints can also be placed on parts. Select the parts on which the constraint should be applied.

Create Constraint Equation

This option allows you to create a multipoint constraint equation of the form: $\sum_j A_j u_j = 0$

where A_j is a real number, and A_1 is non-zero.

A mesh must be loaded in order to define a constraint equation.

Name
the name of the constraint equation.

MPC Set
the set identification number.

LCS
select the Coordinate System from the drop down list.

Node Set ID
specifies the ID number of the Node Set.

The dependent and independent nodes, DOF, and coefficients can be specified.

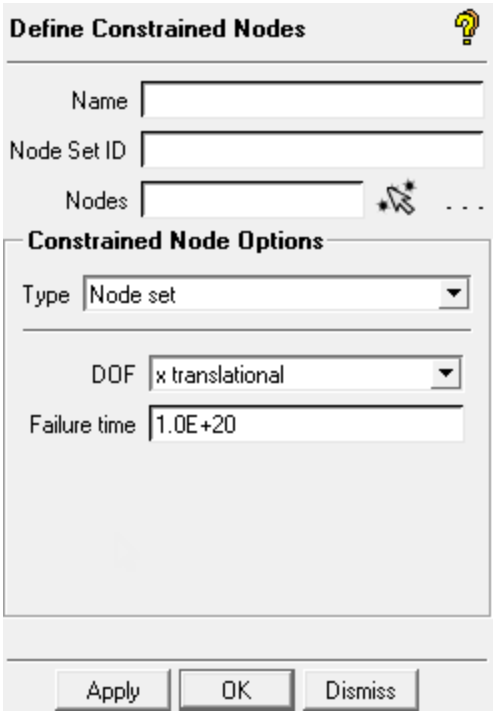
Define Constrained Node Sets

In order to define constrained node sets, a mesh must be loaded.

Note

This option is available for the LS-DYNA, ANSYS, and Autodyn solvers.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The following parameters are specified:

Name
Enter the desired name for your set.

Node Set ID
Enter an identifier for your set.

Nodes
Click the **Select node(s)** icon and then choose the nodes to be constrained in the graphics window.

Type
Choose from **Node set** or **Generalized weld** using the drop-down list. The additional options are dependent on your choice.

Define Contact

The contact option is supported for ANSYS, Abaqus, and LS-DYNA solvers. Nastran does not support contacts. If you need to model contact for Nastran it is recommended to use connectors in place of contact. The options for contact vary by solver.

Contacts can be defined in a variety of ways as described in the following sections:
Automatic Detection
Manual Definition

Automatic Detection

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact Proximity Factor

defines the distance used to evaluate which mesh is within contact with mesh in other parts. For each of the selected parts, mesh between the parts will be evaluated a contact region between the parts will be created.

Note

A mesh file must be loaded for this feature.

Parts

specifies the parts for which contact will be defined. By limiting the number of parts and modifying the contact proximity factor, you can control the contact regions.

Note

Often it is easiest to select all parts.

Create one contact group for all parts

if enabled, one contact group will be created for all the selected parts.

Static Coefficient of Friction

The static coefficient of friction (μ) between two surfaces is defined as the ratio of the tangential force (F) required to produce sliding divided by the normal force between the surfaces (N):

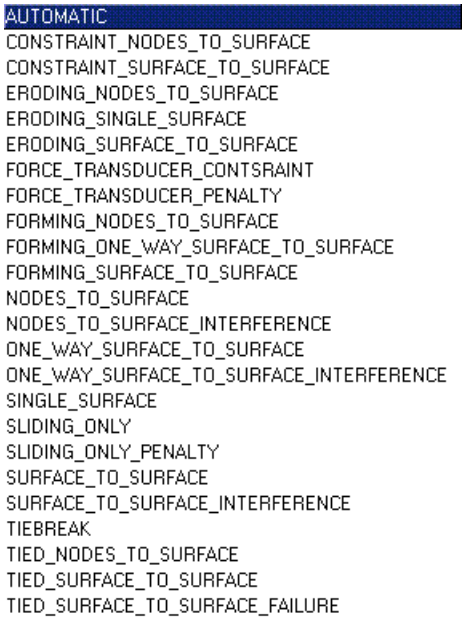
$\mu = F / N$

The static coefficient of friction is used in the contact definition to define the resistance of the two parts under contact.

LS-DYNA Contact Options

The contact cards that are supported are listed in the figure below. For further details, refer to the LS-DYNA manual.

Figure 484: Define Contact Options



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Contact Option
For further details, refer to the LS-DYNA manual.

Dynamic Coefficient of Friction
The dynamic coefficient of friction or kinetic friction is used to define the friction under contact when the two parts are in motion relative to each other (in contrast to the static coefficient of friction).

Manual Definition

Contacts can also be manually defined for shell elements.

Name
the name of the contact, which will appear in the Display Tree.

Contact Surfaces
specifies the surfaces or shell elements that make up the contact region.

Target Surfaces
specifies the surfaces or shell elements that make up the target region.

The remaining parameters are the same as described in Automatic Detection (p. 584)

Define Single Surface Contact

This option allows you to define a single surface contact.

Note

This option is available only for the LS-DYNA solver.

Name
the name of the contact, which will appear in the Display Tree.

Contact Surfaces
specifies the surfaces or shell elements that make up the contact region.

Static Coefficient of Friction
The static coefficient of friction (μ) between two surfaces is defined as the ratio of the tangential force (F) required to produce sliding divided by the normal force between the surfaces (N):

$\mu = F / N$

The static coefficient of friction is used in the contact definition to define the resistance of the two parts under contact.

Dynamic Coefficient of Friction
The dynamic coefficient of friction or kinetic friction is used to define the friction under contact when the two parts are in motion relative to each other (in contrast to the static coefficient of friction).

LS-DYNA Contact Options
There are three types of contact options:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTOMATIC_SINGLE_SURFACE
SINGLE_SURFACE
ERODING_SINGLE_SURFACE

For further details, refer to the LS-DYNA manual.

Define Initial Velocity

This option defines initial nodal point translational velocities using nodal sets. This may also be used for sets in which some nodes have other velocities.

Note

This option is available only for the LS-DYNA solver.

Name
the name of the velocity, which will appear in the Display Tree.

Points
Nodes/Points which are to have defined velocities.

Directional Velocity
specifies the X, Y and Z components of the Translational Velocity.

Rotational Velocity
specifies the X, Y and Z components of the Rotational Velocity.

Define Planar Rigid Wall

This option allows you to define planar rigid walls.

Note

This option is available only for the LS-DYNA solver.

Name
the name of the rigid wall that will be updated in the Display Tree.

Points
Points/0D elements for which you want to define a rigid wall.

Offset
all nodes within this offset distance to the grid wall are included as slave nodes for the rigid wall.

Head Coordinates
coordinates for the head of any outward normal vector, originating on the wall (tail) and terminating in space (head).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tail Coordinates

coordinates for the tail of any outward normal vector, originating on the wall (tail) and terminating in space (head).

Interface Friction Data

- **Type/Coulomb Coeff Value**
  - EQ 0.0, frictionless sliding after contact.
  - EQ 1.0, no sliding after contact.
  - 0.0 < FRIC < 1, Coulomb friction coefficient.
  - EQ 2.0, node is welded after contact with frictionless sliding.
  - EQ 3.0, node is welded after contact with no sliding.
- **Critical Normal Velocity for Weld**

  Critical normal velocity at which the nodes are welded to the wall.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loads

External and internal loading can be applied in a variety of ways. These loads could be gravity (weight of the structure) acting on the parts, pressure, forces or moments acting on a set of geometric or meshed entities, velocities and accelerations, or temperature loading.

The Loads Menu options are described in the following sections.

Create Force
Place Pressure
Create Temperature Boundary Condition

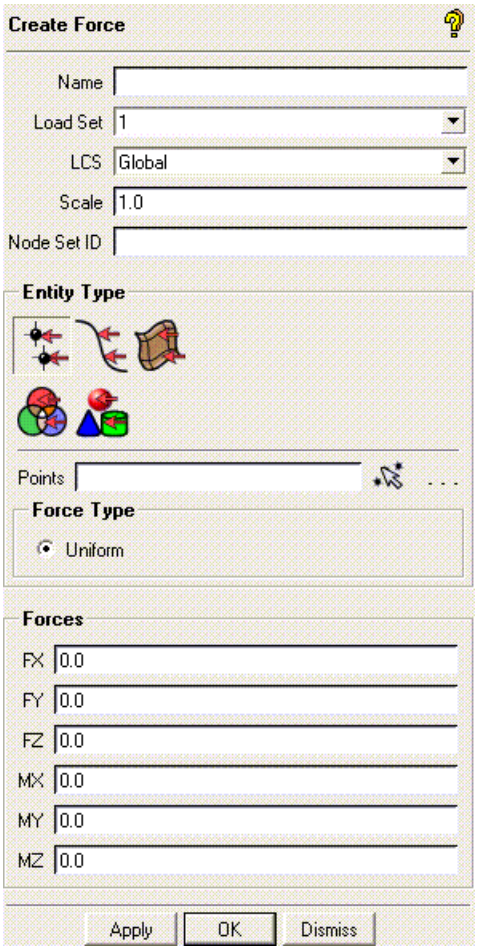
Figure 485: Loads Menu

Create Force

External forces or moments can be applied to different entities.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 486: Create Force Window



Name
Assign a name to the force.

Load Set
Loads are separated into different sets for different properties. Sets are labelled with integer values.

LCS
Select the local coordinate system.

Scale
The scale factor for the value of the force.

Node Set ID
specifies the ID number of the Node Set.

Entity Type
Select the entity type to which the forces or moments will be applied.

Force Type

- **Uniform**

  The force is uniformly distributed on all the nodes of the selected entities.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Total**

  The force is distributed among all the nodes of the selected entities as per FEA concepts.

Forces

Enter the values of the forces and moments for the corresponding directions.

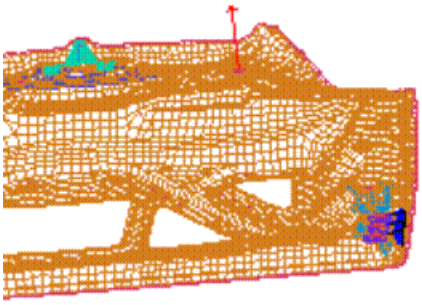
Note

Applied forces will be represented by a blue arrow, and moments by a brown arrow. If both are applied at any location then a brown arrow will represent both the force and moment.

Create Force on Point

External forces or moments can be applied to geometric points or nodes with this option.

Figure 487: Example of Externally Applied Forces

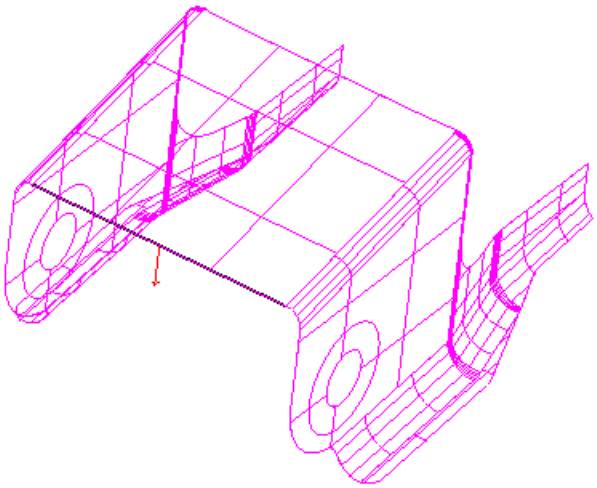


Create Force on Curve

Forces or moments can also be applied on geometric lines or curves before meshing, as shown in the example below. Force or moment values will be enforced on all the nodes created on the specified curves after meshing.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 488: Example of Edge Loading



Create Force on Surface

Forces or moments can also be applied on surfaces before meshing.

Create Force on Subset

Forces or moments can be applied on subsets of entities with the same options as described above.

Note

The subset must already be created.

Create Force on Part

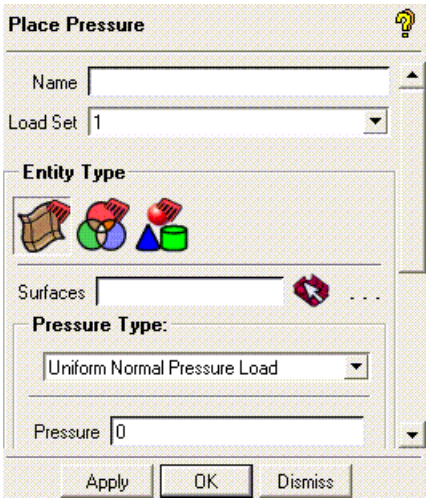
Forces or moments can be applied on parts with the same options as described above.

Place Pressure

Pressures can be defined to apply to surfaces, subsets, or parts. The metric unit of pressure is N/mm^2, and the pressure is applied to all elements on the entity once meshing is done.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 489: Place Pressure Window



Name
Assign a name to the pressure.

Load Set
Loads for different properties are grouped by sets. Sets are labelled by integer values.

Entity Type
Select the entities to which the pressure is to be applied.

Pressure Type

- **Uniform Normal Pressure Load**

  Enter the magnitude of the pressure that will be uniformly applied.

- **General Pressure Load**

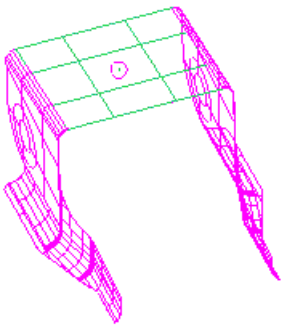
  Enter the parameters to define the General Pressure Load.

Place Pressure on Surface

Pressure can be applied to surfaces or shell elements. When writing out to an output file, the pressure will be applied to the shell elements or the attached faces of the solid element, depending on what solver is chosen and the type of pressure. The direction the pressure is applied is based on the surface normals of the shell elements, or is applied into the volume for volume mesh. A circle symbol will be displayed on a surface to indicate application of pressure, as shown in the figure below.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 490: Pressure on Surface Example



Place Pressure on Subset

Pressures to a subset of geometric or mesh entities are defined with the same options. The subset must already be created.

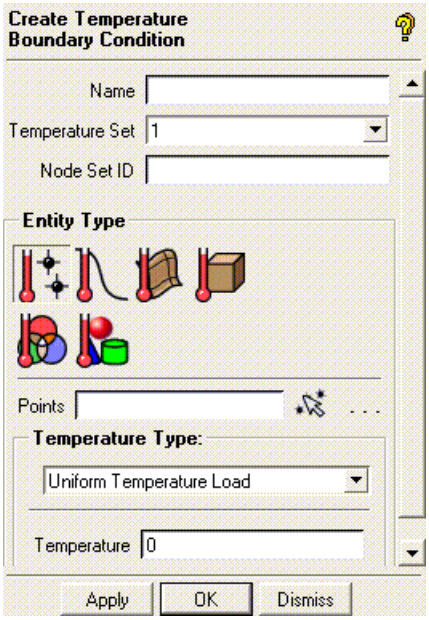
Place Pressure on Part

Pressures to a subset of geometric or mesh entities are defined with the same options.

Create Temperature Boundary Condition

The metric unit of temperature used is °K (Kelvin scale).

Figure 491: Create Temperature Boundary Condition Window



Name

Name of the Temperature Set.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Set
Set number for the defined Temperature.

Node Set ID
specifies the ID number of the Node Set.

Entity Type
Select the entities to which the temperature boundary condition is to be applied.

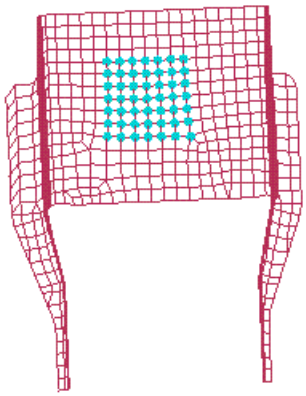
Temperature Type
This option applies to points only. The temperature load will be applied uniformly.

Temperature
The value of the temperature acting on the selected entities.

Temperature on Point

Temperatures can be applied to curves. In the following example, thermal boundary curves are indicated by highlighted dots on the nodes.

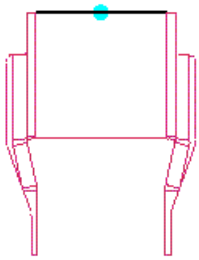
Figure 492: Thermal Loading on Nodes



Temperature on Curve

Temperatures can be applied to curves. In the following example, thermal boundary curves are indicated by highlighted dots on the curves.

Figure 493: Temperature on Curves Boundary Condition Example

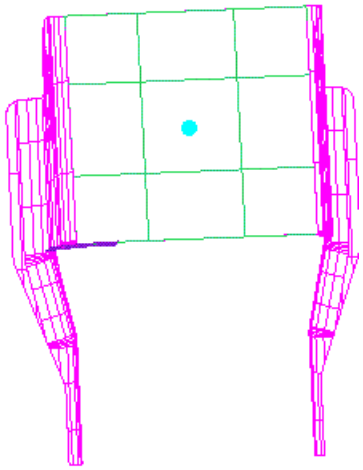


ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature on Surface

Temperatures can be applied to surfaces. In the following example, thermal boundary conditions are indicated by highlighted dots on the surfaces.

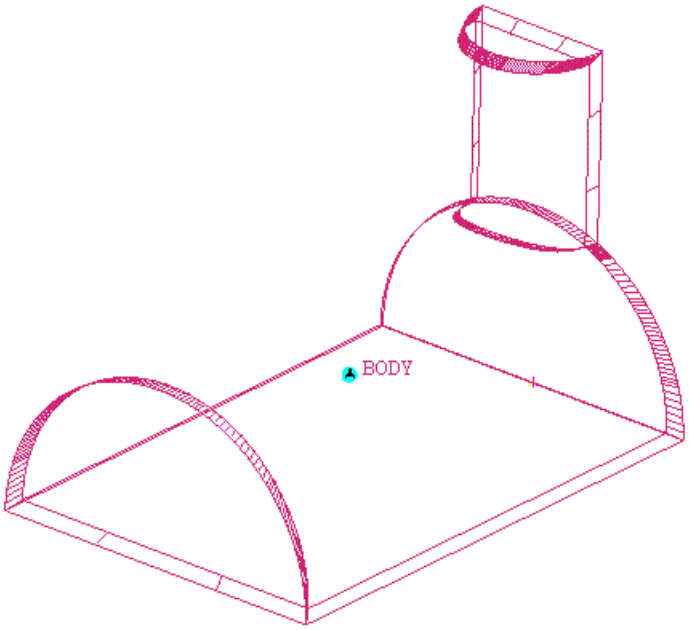
Figure 494: Surface Temperature Boundary Condition Example



Temperature on Body

Temperatures can be applied to bodies as well. In the following example, thermal boundary conditions are indicated by highlighted dots on the bodies.

Figure 495: Body Temperature Boundary Condition Example



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature on Subset

Temperatures can be also be applied to subsets. The subset must already be created.

Temperature on Part

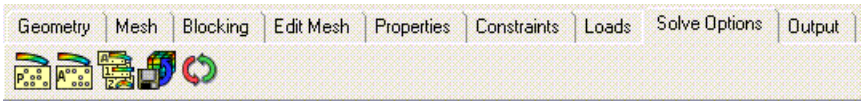
Temperatures can be also be applied to parts.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solve Options

Figure 496: Solve Options Menu



After setting up the different loading conditions on the geometry and mesh of a model, the **Solve Options** Menu enables you to set specific options for your analysis.

If you selected a **Common Structural Solver** in the **Solver Setup** DEZ, (see **Settings > Solver**), your selection will be visible in the **Method** drop-down list. If not, choose a **Solver** method at this time. Your choices are NASTRAN, ANSYS, LS-DYNA, Abaqus, or Autodyn.

Note

To run ANSYS through the FEA functionality, you need to add the following environment variables:

ANSYS_EXEC_PATH and ANSYSnn_PRODUCT.

The variable ANSYS_EXEC_PATH has to point to the “ansys.exe” file in the ANSYS installation area.

The variable ANSYSnn_PRODUCT has to be set for the ANSYS Product through which analysis will be done. Here, nn is the ANSYS version. The ANSYS product list is given below.

License Feature	Product Description
ane3fl	ANSYS/Multiphysics
ANSYS	ANSYS/Mechanical\ U
ane3	ANSYS/Mechanical/Emag\ 3D
Anfl	ANSYS/Mechanical/FLOTRAN
ane3flds	ANSYS/Multiphysics/LS-DYNA
Ansysds	ANSYS/Mechanical/LS-DYNA
ane3ds	ANSYS/Mechanical/Emag\ 3D/LS-DYNA
Anflds	ANSYS/Mechanical/FLOTRAN/LS-DYNA
ane3fldp	ANSYS/Multiphysics/DYNAPrepPost
Ansysdp	ANSYS/Mechanical/DYNAPrepPost
ane3dp	ANSYS/Mechanical/Emag\ 3D/DYNAPrepPost
Anfldp	ANSYS/Mechanical/FLOTRAN/DYNAPrepPost
Struct	ANSYS/Structural\ U

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

License Feature	Product Description
ste3	ANSYS/Structural/Emag\ 3D
Stfl	ANSYS/Structural/FLOTRAN
ste3fl	ANSYS/Structural/Emag\ 3D/FLOTRAN
Structds	ANSYS/Structural/LS-DYNA
ste3ds	ANSYS/Structural/Emag\ 3D/LS-DYNA
Stflds	ANSYS/Structural/FLOTRAN/LS-DYNA
Ste3flds	ANSYS/Structural/Emag\ 3D/FLOTRAN/LS-DYNA
Structdp	ANSYS/Structural/DYNAPrepPost
Ste3dp	ANSYS/Structural/Emag\ 3D/DYNAPrepPost
Stfldp	ANSYS/Structural/FLOTRAN/DYNAPrepPost
Ste3fldp	ANSYS/Structural/Emag\ 3D/FLOTRAN/DYN-APrepPost
Prf	ANSYS/Professional
prfe3	ANSYS/Professional/Emag\ 3D
Prffl	ANSYS/Professional/FLOTRAN
prfe3fl	ANSYS/Professional/Emag\ 3D/FLOTRAN
ane3fl1	ANSYS/Multiphysics\ 1
ane3fl2	ANSYS/Multiphysics\ 2
ane3fl3	ANSYS/Multiphysics\ 3
Ansysuh	ANSYS/University\ High
Ansysul	ANSYS/University\ Low
Ansysrf	ANSYS/ResearchFS
struct1	ANSYS/Structural1
struct2	ANSYS/Structural2
struct3	ANSYS/Structural3
Mpba	ANSYS/Multiphysics\ Batch
Meba	ANSYS/Mechanical\ Batch
Prba	ANSYS/Professional\ Batch
Stba	ANSYS/Structural\ Batch
Mpbach	ANSYS/Multiphysics\ Batch\ Child
Mebach	ANSYS/Mechanical\ Batch\ Child
Prbach	ANSYS/Professional\ Batch\ Child
Stbach	ANSYS/Structural\ Batch\ Child
Debach	ANSYS/DesignSpace\ Batch\ Child

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

License Feature	Product Description
Profea	ANSYS/ProFEA

Note

The LS-DYNA deck does not handle units. The user is responsible to use consistent units for all data.

Since Autodyn uses LS-DYNA data, the same caveat applies.

The following sections will address the **Solve Options** available. Each section is further split based on Solver selection.

Setup Solver Parameters
Setup Analysis Type
Setup a Subcase
Write/View Input File
Submit Solver Run

Setup Solver Parameters

Use this option to configure the parameters for the selected Solver.

The **Setup Solver Parameters** window has different options for different Solvers, as explained below.

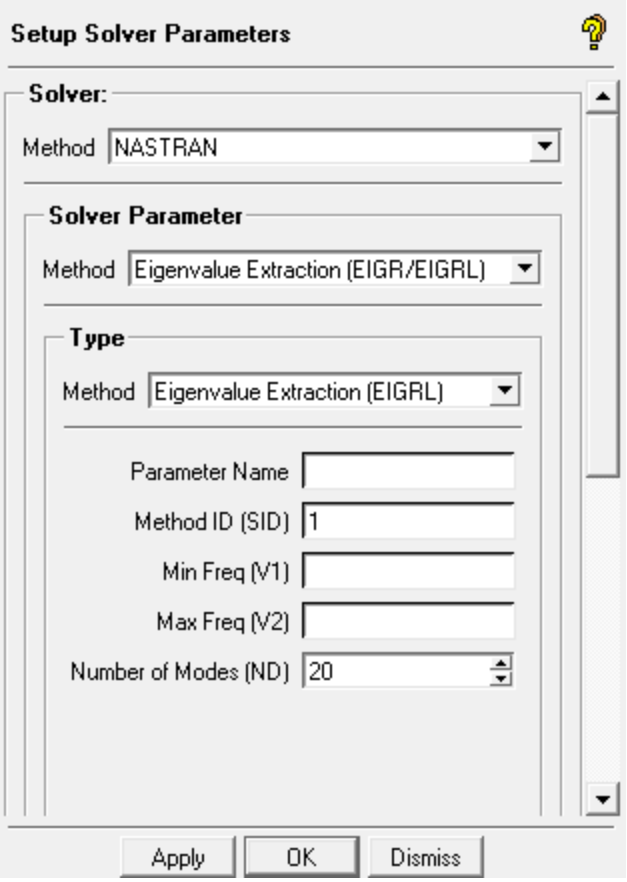
NASTRAN Setup Solver Parameters
ANSYS Setup Solver Parameters
LS-DYNA Setup Solver Parameters

Note

There are no additional parameters for Abaqus Solver or Autodyn Solver setup

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NASTRAN Setup Solver Parameters



The different types of analysis that can be performed are: *Linear Static Analysis*, *Modal Analysis* and *Legacy analysis*. Each analysis requires various parameters and inputs, which are explained in the following sections.

Eigenvalue Extraction (EIGR / EIGRL)

- **Method ID (SID)**

  Set ID number (Unique integer > 0)

- **V1, V2**

  For vibration analysis, the frequency range of interest.

- **Number of Modes (ND)**

  Number of frequencies desired.

Buckling Analysis (EIGB)

- **Method ID (SID)**

  Set ID number (Unique integer > 0)

- **SINV**

  Enhanced inverse power method.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **INV**

  Inverse power method.

- **Lower / Upper Eigenvalues (L1, L2)**

  The Eigenvalue range of interest. Eigenvalues are the factors by which the pre-bucking state of stress is multiplied to produce buckling in the shape defined by the corresponding Eigenvector.

- Desired Number of Positive / Negative Roots (NDP, NDN)

- **NORM**

  Method for normalizing eigenvectors. The **MAX** option (default) normalizes eigenvectors to the unit value of the largest component in the analysis set. The **POINT** option normalizes eigenvectors to the unit value of the component defined in the **G** and **C** fields, where G is the Grid or scalar point ID, and C is the Component number. The value for NORM defaults to MAX if the defined component is zero.

Nonlinear Static Analysis Control (NLPARM)

- **Number of Increments (NINC)**

  0<Integer<1000, Default=10.

- **Incremental Time Intervals (DT)**

  Incremental time interval for creep analysis. The unit of DT must be consistent with units used for the CREEP entry that defines its characteristics.

- **KMETHOD**

  The stiffness update strategy.

- **Number of Iterations (KSTEP)**Number of iterations before the stiffness update for the ITER method. The stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.

- **Max Iterations (MAXITER)**The limit on the number of iterations for each load increment.

- **CONV**

  Flags to select convergence criteria. U=Displacement error, P=Load equilibrium error, W=work error and the error tolerances (**EPSU**, **EPSP**, and **EPSW**) define the convergence criteria.

- **INTOUT**

  Intermediate output flag. Controls the output requests for displacements, element forces and stresses, etc.

- **Max Divergence (MAXDIV)**

  The limit on the probable divergence condition per iteration before the solution is assumed to diverge. (Integer not equal to 0: Default=3)

- **Manimum Number of Quasi-Newton (MAXQN)**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Newton correction vectors to be saved in the database.

- **Max Number of Line Search (MAXLS)**

  Maximum number of line searches allowed for each iteration.

- **Fraction of Effective Stress (0.0–1.0) (FSTRESS)**

  Fraction of effective stress used to limit the sub-increment size in the material routines.

- **Line Search Tolerance (0.01–0.9) (LSTOL)**

  Line search tolerance.

- **Max Number of Bisections (MAXBIS)**

  Maximum number of bisections allowed for each load increment.

- **Max Ratio (1.0–40.0) (MAXR)**

  Maximum ratio for the adjusted arc length increment relative to the initial value.

- **Max Value of Incremental Rotation (RTOLB)**

  Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection.

Transient Time Step (TSTEP)

Defines the step intervals at which a solution will be generated, and the output in transient analysis.

- **Number of Time Steps**

  Number of time steps of value DTi (integer greater than or equal to 1)

Frequency (FREQ / FREQ1 / FREQ2)

- **Frequency List (FREQ)**

  – Number of Frequencies

    Defined number of frequencies.

- **Frequency List (FREQ1)**

  The Frequency List (FREQ1) options define a set of frequencies to be used in the solution of frequency response problems, specified by the **First Frequency (F1)**, the **Frequency Increment (DF)**, and the **Number of Frequency Increments (NDF)** desired.

- **Frequency List (FREQ2)**

  The Frequency List (FREQ2) options define a set of frequencies to be used in the solution of frequency response problems, specified by the **First Frequency (F1)**, the **Frequency Increment (DF)**, and the **Number of Logarithmic Increments (NF)**.

Dynamic Load (DAREA / DELAY / DPHASE)

- **Load Scale Factor (DAREA)**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOAD entries.

– **Pi**

Grid, extra or scalar point identification number. (Integer >0)

– **Ci**

Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).

– **Ai**

Scale (area) factor.

- **Dynamic Load Time Delay (DELAY)**

– **Pi**

Grid, extra or scalar point identification number. (Integer >0)

– **Ci**

Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).

– **Ti**

Time delay for designated point Pi and component Ci.

- **Dynamic Load Phase Lead (DPHASE)**

Defines the phase lead term theta in the equation of the dynamic loading function.

– **Pi**

Grid, extra or scalar point identification number. (Integer >0)

– **Ci**

Component number (Integer 1 through 6 for grid point, blank or 0 for extra scalar point).

– **THi**

Phase lead theta in degrees.

Dynamic Excitation (TLOAD/RLOAD)

- **Transient Response (TLOAD1)**

Defines a time dependent dynamic load or enforced motion, for use in transient response analysis.

– **DAREA**

Identification number of DAREA set or a thermal load set that defines enforced acceleration using large mass or SPC/SPCD data.

– **DELAY**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Identification number of delay entry set.

– **EXCITATION TYPE**

Defines the type of dynamic excitation.

– **TABLEDi**

Identification number of TABLEDi entry.

- **Transient Response (TLOAD2)**

  Includes the same parameters as Transient Response (TLOAD1), in addition to the parameters: **Time Constants (T1, T2)**, **Frequency (F)**, **Phase Angle (P)**, **Exponential Coef. (C)**, and **Growth Coef. (B)**.

- **Frequency Response (RLOAD1)**

  Defines a frequency dependent dynamic excitation for use in frequency response problems.

- **Frequency Response (RLOAD2)**

  Includes the same parameters as Frequency Response (RLOAD1).

Dynamic Load Combination (DLOAD)

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response, or TLOAD1 or TLOAD2 entries for transient response.

- **Global Scale Factor (S)**

  The Global Scale Factor.

- **Number of loads**

  The number of Load Set ID numbers of RLOAD1, RLOAD2, TLOAD1, and TLOAD2.

Static Load Combination (LOAD)

Defines a static load as a linear combination of load sets.

- **Global Scale Factor (S)**

  The Global Scale Factor.

- **Number of loads**

  The number of Load Set ID numbers used to define the LOAD.

Single-Point Constraint Combination (SPCADD)

Defines a single-point constraint set as the combination of other defined single-point constraint sets.

Multi-Point Constrant Combination (MCADD)

Defines a multipoint constraint set as combination of other multippoint constraint sets.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Grid Point Temperature (TEMPD)

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

Acceleration or Gravity Load (GRAV)

Defines acceleration vectors for gravity or other acceleration loading.

- **LCS**

  Identify the local coordinate system.

- **MB**

  Indicates whether the local coordinate system (LCS) is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are defined relative to the assembly basic coordinate system, which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data section is rotated or mirrored.

- **Scale Factor**

  Scale factor of the acceleration vector.

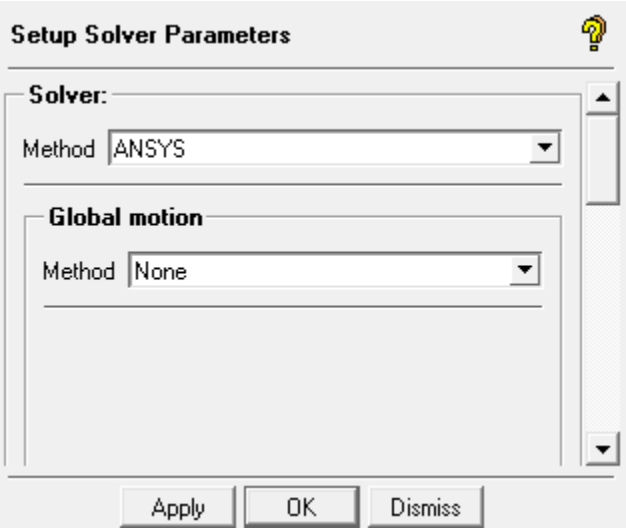
- **Vector Components**

  Acceleration vector components measured in the specified coordinate system.

Include File (INCLUDE)

Inserts an external file (*.dat, *.nas, *.bdf) into the input file using an INCLUDE statement.

ANSYS Setup Solver Parameters



Use the drop down list to select the **Method** of Global motion that is to be specified, if any.

Acceleration or Gravity

- **Parameter Name**

  The name of the load.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scale Factor**

  Scale factor of the acceleration vector.

- **Direction**

  Specify the acceleration vector components.

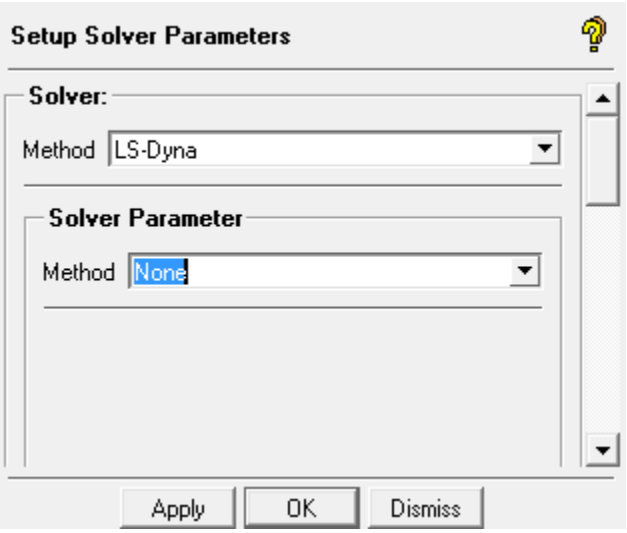
Rotational Velocity

The X, Y, and Z components can be specified. Also, the Spin Softening Key can be decreased or not modified.

Rotational Acceleration

The X, Y, and Z components can be specified.

LS-DYNA Setup Solver Parameters



Base Acceleration or Gravity Load

Defines acceleration vectors for gravity or other loading.

Note

Only one gravity vector can be defined for the LS-DYNA file.

- **Parameter Name**

  The name of the load.

- **Scale Factor**

  Scale factor of the acceleration vector.

- **Direction**

  Specify the acceleration vector components.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup Analysis Type

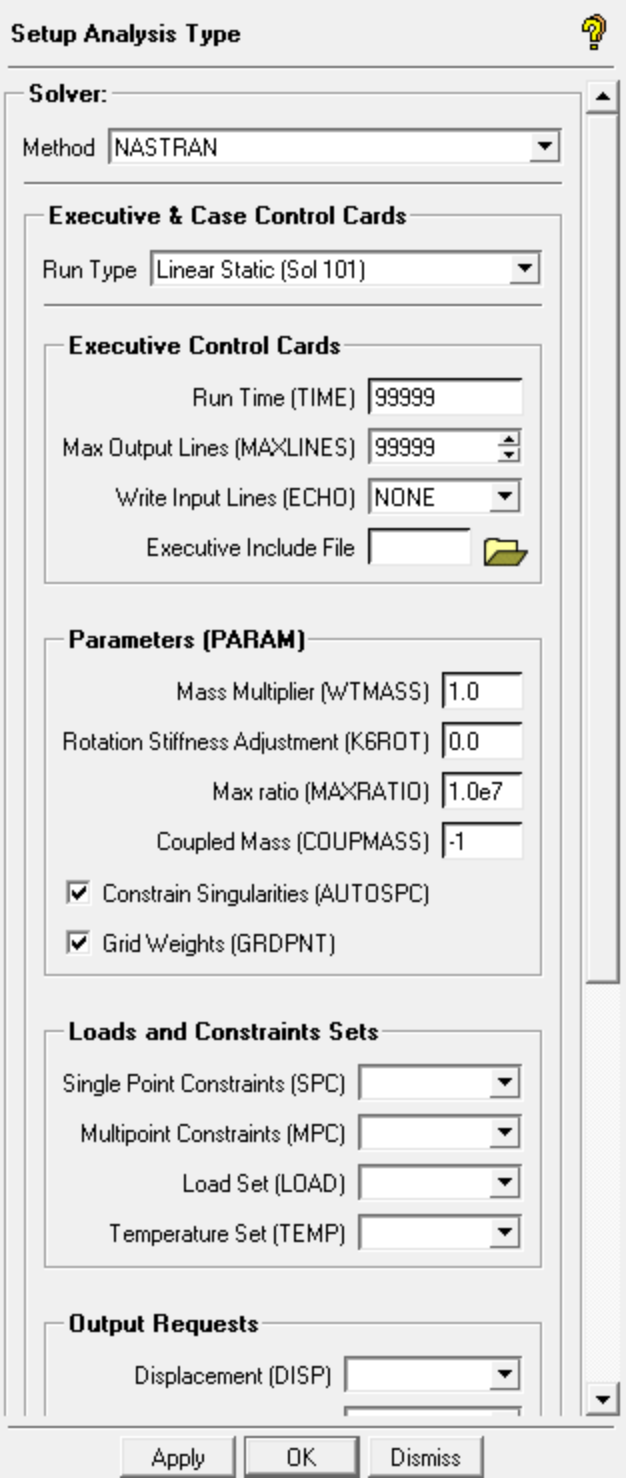
This option allows you to setup the parameters for the different types of analysis.

The following sections describe the parameters that are specific to each analysis type.

NASTRAN Setup Analysis Type
ANSYS Setup Analysis Type
LS-DYNA Setup Analysis Type
Abaqus Setup Analysis Type
Autodyn Setup Analysis Type

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NASTRAN Setup Analysis Type



Run Time (TIME)

The maximum allowable execution time in CPU minutes. By default, it is set to *99999*. You can increase this if necessary.

Max Output Lines (MAXLINES)

The maximum number of output lines to the solver. In any run, if MAXLINES exceeds the set value (*99999* is the default), then the program will terminate.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write Input Lines (ECHO)
Controls the Input file for the Nastran solver. The following options are available:

- **NONE**

  Suppresses printing of the Bulk Data into the Nastran Input file.

- **SORT**

  Writes the Bulk Data section in small field format as well as sorted alphabetically into the Nastran input file.

- **UNSORT**

  Writes the Bulk Data section exactly as it is input.

- **BOTH**

  Write both the SORT and UNSORT formats.

- **PUNCH**

  Punches the Bulk Data to an ASCII file.

Case Control Cards

Output Title (TITLE)
To specify a title that will appear on the first heading line of each output page.

Output SubTitle (SUBTITLE)
To specify a subtitle that will appear on the second heading line for each output page.

Output Label (LABEL)
To specify a label that will appear on the third heading line of each output page.

Linear Static Analysis (Sol 101)

Mass Multiplier (WTMASS)
This value will be multiplied to the structural mass matrix when the mass of the structure is calculated.

Rotation Stiffness Adjustment (K6ROT)
It specifies the stiffness to be added to the normal rotation for Shell and Tri elements. The default is 0, but you can change its value between 1 to 100 to suppress singularities for geometric nonlinear analysis.

Max Ratio (MAXRATIO)
The default value for the MAXRATIO is 1.0E7.

Coupled Mass (COUPMASS)
The default value is -1.

Constrain Singularities (AUTOSPC)
AUTOSPC specifies the action to take when singularities exist in the stiffness matrix. If enabled, the singularities will be constrained automatically.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Grid Weights (GRDPNT)
This option will execute the grid point weight generator. It specifies the identification number of the grid point to be used as a reference point.

Single Point Constraints (SPC)
Sets this value (positive integer) for the solution. It will set the same number of Single Point Constraint in the input data form.

Load Set (LOAD)
Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set (TEMP)
Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Output Requests
These output requests (Displacement, Stress, Strain, Element Strain Energy) can be set to ALL or NONE.

Modal (Sol 103)

The Parameters and Output Requests data that is needed to perform Modal analysis is described in Linear Static Analysis (Sol 101) (p. 611).

Buckling Analysis (Sol 105)

The Parameters and Output Requests data that is needed to perform Buckling analysis is described in Linear Static Analysis (Sol 101) (p. 611).

Nonlinear Static (Sol 106)

The Parameters and Output Requests data for performing Nonlinear Static analysis is described in Linear Static Analysis (Sol 101) (p. 611).

Direct Frequency Response (Sol 109)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 611). The additional data needed is described below.

Parameters (PARAM)

- **Structural Coefficient (G)**

  This value is equal to C/C_0 multipled by a factor of 2.

Case Control Cards

- **Dynamic Load**

  Select a dynamic load to be applied in a transient or frequency response problem.

- **Frequency**

  Select a frequency to be applied in a transient or frequency response problem.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direct Transient Response (Sol 109)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 611). The additional data needed is described below.

Parameters (PARAM)

- **Structural Coefficient (G)**

  This value is equal to C/C_0 multipled by a factor of 2.

- **Structural Coefficient (W3)**

  The default value is 0.

Output Requests

- **Velocity**

  Requests the form and type of velocity vector output.

Case Control Cards

- **Dynamic Load**

  Select a dynamic load to be applied in a transient or frequency response problem.

- **Frequency**

  Select a frequency to be applied in a transient or frequency response problem.

Modal Frequency Response (Sol 111)

The Parameters and Output Requests data that is needed to perform this analysis is described in Linear Static Analysis (Sol 101) (p. 611). The additional data needed is described below.

Output Requests

- **Velocity**

  Requests the form and type of velocity vector output.

- **Acceleration**

  Requests the acceleration output.

Case Control Cards

- **Dynamic Load**

  Select a dynamic load to be applied in a transient or frequency response problem.

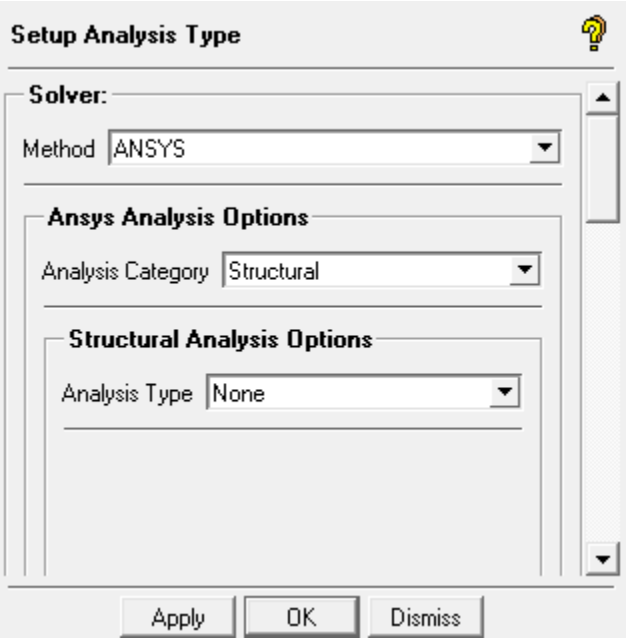
- **Frequency**

  Select a frequency to be applied in a transient or frequency response problem.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Structural Damping (SDAMPING)**

ANSYS Setup Analysis Type



Two different categories of analysis are supported for the ANSYS solver: **Structural** and **Thermal** Analysis. The following sections describe the parameters for each type of analysis.

Structural Analysis

For Structural Static analysis, select the type of solver:

- **Auto**
  - **Number of Substeps**

    Number of substeps to be used for this load step (the time step size or frequency increment).
  - **Large Deformation Key**

    If **Ignore** is selected, than it will not include large deflection effects in a static or full transient analysis. For the **Include** option, it will be incorporated in the analysis.
- **Direct**

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
- **PCG**

  This is a Pre-conditioned Conjugate Gradient iterative equation solver. It requires less disk space and is faster for large models.

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.
  - **Tolerance**

    Solver tolerance value (defaults to 1.0e-8).

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Convergence multiplier**

Multiplier user to control the maximum number of iterations performed during a convergence calculation. The recommended range for the multiplier is $1.0 \leq$ multiplier ≤ 3.0.

The Structural Modal Analysis, the following parameters can be defined for the **Mode Extraction Method**.

Block Lanczos
This is a modal analysis method that is supported in the FEA functionality of ANSYS ICEM CFD.

Number of Modes to Extract
Number of modes to extract for the analysis.

Beginning (Lower End) Frequency
Beginning or lower end of the range of frequency. It also represents the first shift point for the eigenvalue iterations.

Ending (Upper End) Frequency
Ending or upper end of the range of frequency. The default value is 1.0E8.

Mode Shape Normalization Key
If set to **OFF**, it normalizes the mode shape to the mass matrix. If set to **ON** , it normalizes the mode shapes to unity instead of to the mass matrix.

Constraint Equation Processing Key
There are four options for different constraint equations to be used during analysis: **Default**, **Quick Lagrange**, **Accurate Lagrange**, and **Direct Elimination**.

Element Calculation Key
Number of modes to expand and write for a modal analysis. If **OFF** is selected, then it will not calculate element results and reaction forces. If **ON** is selected, along with the nodal degree of freedom, it also calculates element results and reaction forces.

Thermal

For Thermal Static analysis, select the type of solver:

- **Auto**

  – **Number of Substeps**

  Number of substeps to be used for this load step (the time step size or frequency increment).

  – **Large Deformation Key**

  If **Ignore** is selected, than it will not include large deflection effects in a static or full transient analysis. For the **Include** option, it will be incorporated in the analysis.

- **Direct**

  Enter the **Number of Substeps** and select the **Large Deformation Key** option.

- **ICCG**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

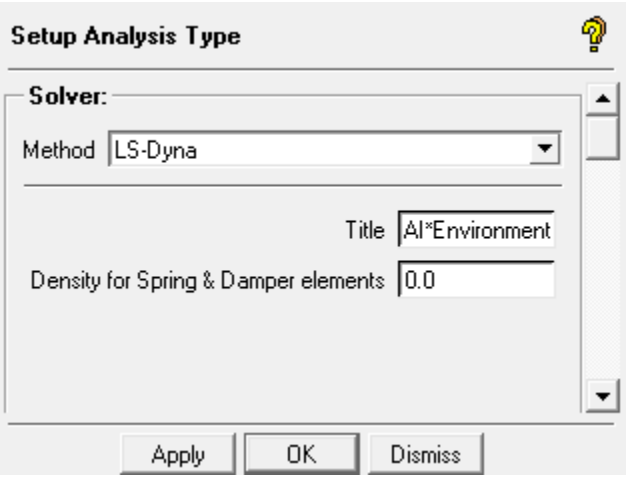
The incomplete Cholesky Conjugate gradient (ICCG) algorithm is a commonly used iterative method for solving large sparse systems of equations.

Enter the **Number of Substeps** and select the **Large Deformation Key** option.

– **Tolerance**

Solver tolerance value (defaults to 1.0e-8).

LS-DYNA Setup Analysis Type



Title

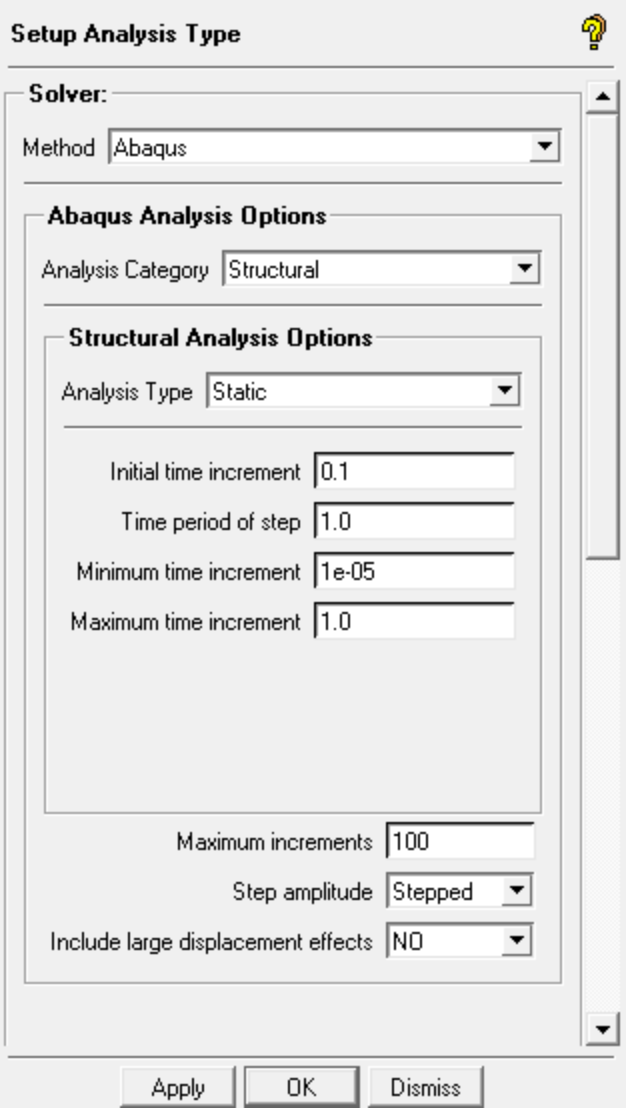
Enter the title of the LS-DYNA solver run.

Density for Spring & Damper Elements

Define the density for the Spring and Damper elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Abaqus Setup Analysis Type



For Abaqus, two **Analysis Type**s are supported: **Static** and **Natural Frequency Extraction**.

For Static analysis, the Initial time increment, Time period of each step, Minimum and Maximum time increments need to be specified.

The Natural Frequency Extraction procedure performs eigenvalue extraction to calculate natural frequencies and the corresponding mode shape of a system. It includes the initial stress and load stiffness effect, and is a linear perturbation procedure. The following parameters pertain to the Natural Frequency Extraction type of analysis.

Eigensolver

There are two types of Eigensolvers, **Lanczos** and **Subspace** Iteration. Lanczos is faster because you can extract a large number of eigen modes (e.g., airframes, piping systems and building skeletons). The subspace iteration method may be faster when only a few (less then 20) eigen modes are needed.

Number of Modes to Extract

Specify the number of modes to be extracted.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beginning (minimum) and Ending (maximum) Frequency
Specify the beginning and ending frequencies.

Shift point
Specify the shift point.

Maximum iterations
Specify the maximum number of iterations.

Normalized By
Specify whether it is to be normalized by Mass Matrix or Large Displacement.

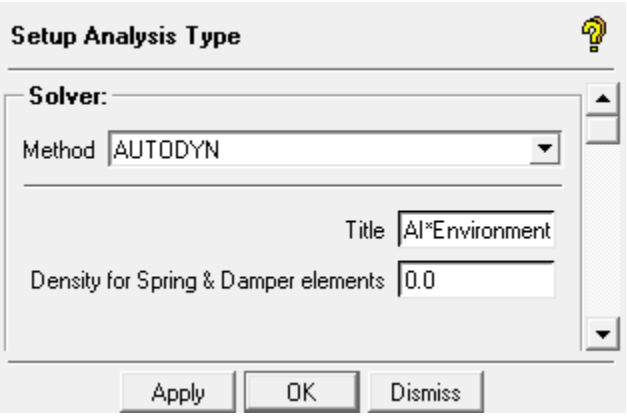
The following parameters pertain to both types of analysis.

Maximum increments
Specify the maximum number of increments.

Step amplitude
Specify whether to use Stepped or Ramped amplitude.

Include large displacement effect
Specify whether to include the effect of large displacement.

Autodyn Setup Analysis Type



Title
Enter the title of the Autodyn solver run.

Density for Spring & Damper Elements
Define the density for the Spring and Damper elements.

Setup a Subcase

The **Setup a Subcase** DEZ allows you to create or modify subcases for analysis.

The details and choices for each Solver **Method** are described in the following sections.
NASTRAN Setup a Subcase
ANSYS Setup a Subcase

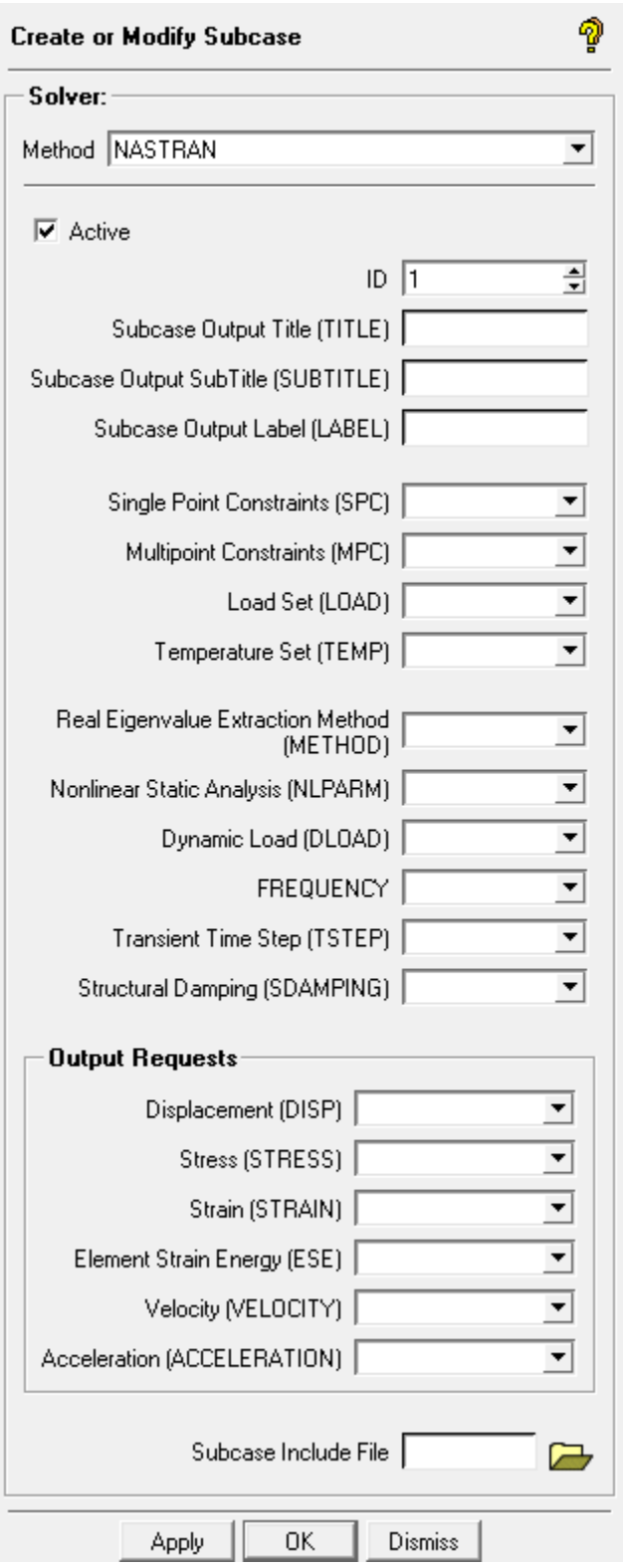
ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Abaqus Setup a Subcase

Note

Subcases are not supported for LS-DYNA or Autodyn.

NASTRAN Setup a Subcase



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active
Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

ID
Identification number for the subcase.

Subcase Output Label (LABEL)
The output label for the subcase.

Single Point Constraints (SPC)
Sets this value (positive integer) for the solution. It will set the same number of Single Point Constraints in the input data form.

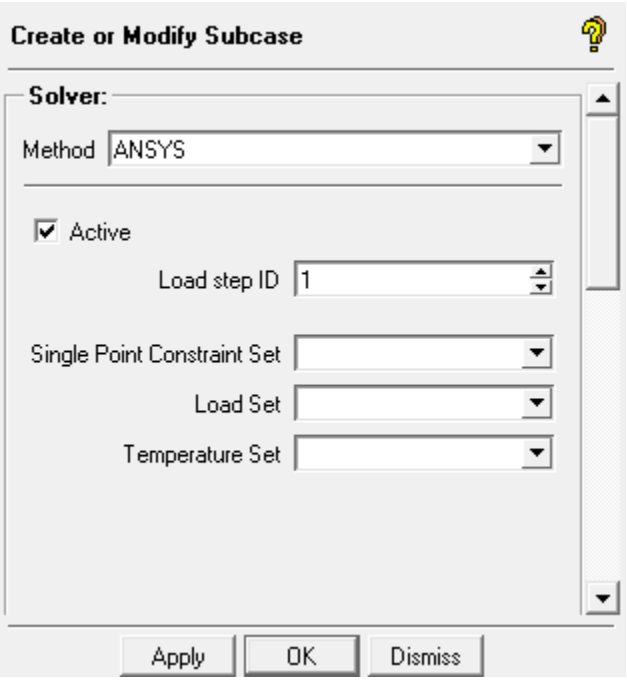
Load Set (LOAD)
Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set (TEMP)
Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Real Eigenvalue Extraction Method (METHOD)
To create a subcase for Buckling Analysis, enter either the INV or SINV method.

Output Requests
These output requests (Displacement, Stress, Strain, Element Strain Energy, Velocity, Acceleration) can be set to ALL or NONE.

ANSYS Setup a Subcase



ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active
Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

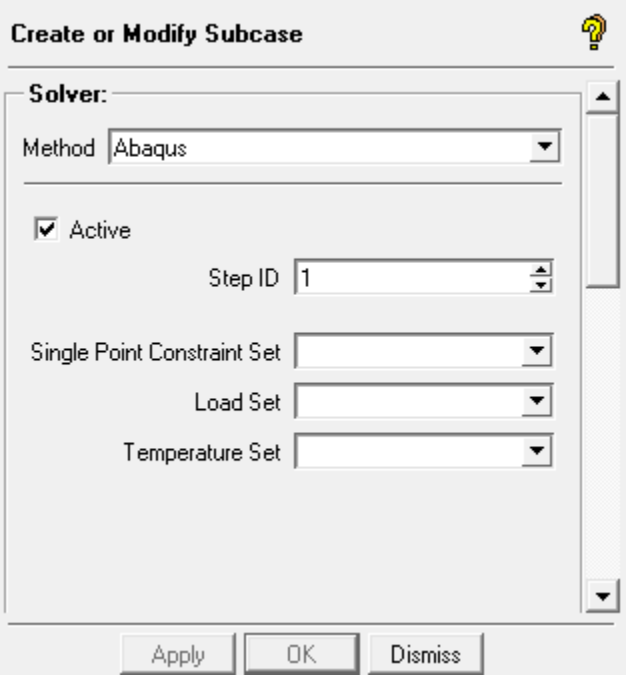
Load step ID
Identification number for the load step.

Single Point Constraint Set
Assigns this set (positive integer) to the solution. It will also set the same number of Single Point Constraints in the input data form.

Load Set
Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set
Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Abaqus Setup a Subcase



Active
Multiple subcases can be created, but only those toggled **Active** will be written out for the solution. This allows for multiple solution types to be written from the same project without having to delete or create subcases.

Load step ID
Identification number for the load step.

Single Point Constraint Set
Assigns this set (positive integer) to the solution. It will also set the same number of Single Point Constraints in the input data form.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Load Set

Assigns an external static load set (positive integer) to the solution. It will set the same number of Load Constraints in the input data form.

Temperature Set

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

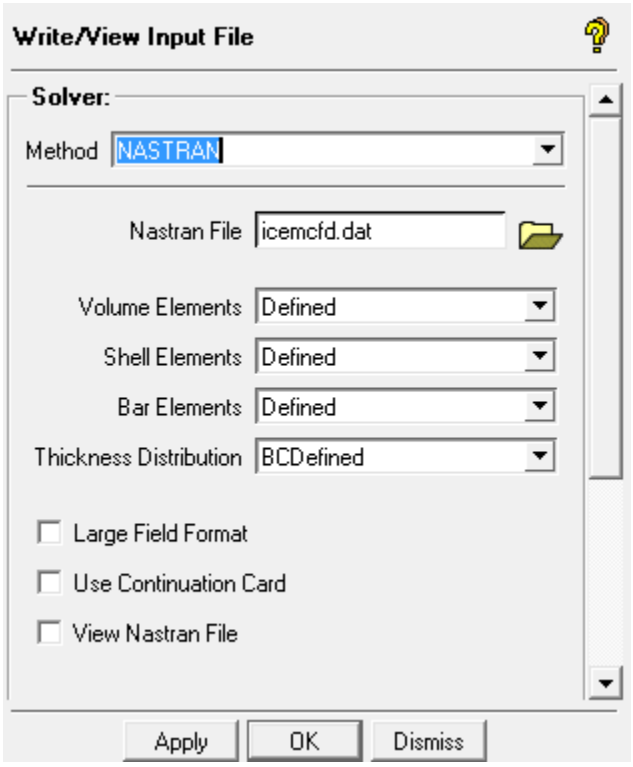
Write/View Input File

The **Write/View Input File** DEZ allows you to choose options for creating or editing the solver input file.

The details and choices for each Solver **Method** are described in the following sections.

NASTRAN Write/View Input File
ANSYS Write/View Input File
LS-DYNA Write/View Input File
Abaqus Write/View Input File
Autodyn Write/View Input File

NASTRAN Write/View Input File



Nastran File

Specify the path and name for the nastran file (*.dat) that will be created after the Nastran run is completed.

Volume, Shell, Bar Elements

The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

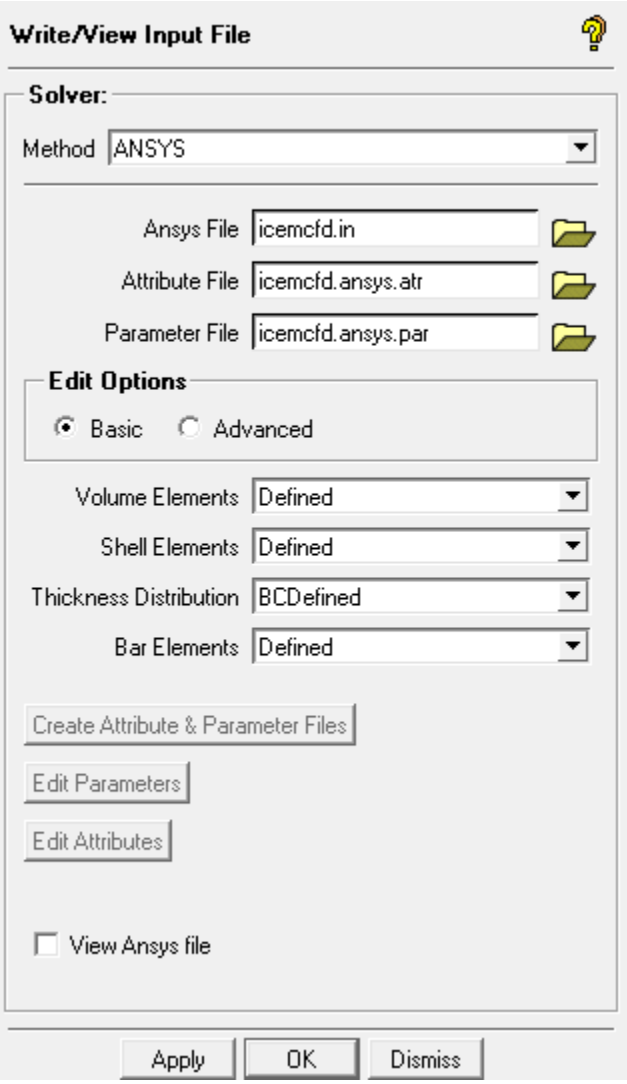
Thickness Distribution
Select from **BC Defined**, **Distributed**, or **Averaged**.

Large Field Format
If enabled, the Nastran input file will be written in Large Field Format. The default Small Format contains 8 characters per field. The Large Field Format writes out the GRID data with 16 characters per field, which is useful when more accurate GRID data is needed.

Use Continuation Card
If enabled, the Nastran data will be output using continuation card. The default is OFF, but some solvers that use Nastran input may still require continuation cards.

View Nastran File
If enabled, after the Nastran input file is written, it will open it in the default editor to be viewed. You can edit this file before submitting it for the Nastran Run.

ANSYS Write/View Input File



ANSYS File
Select the ANSYS Input file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Attribute File
Select the Attribute file which contains information about all the parts and material properties.

Parameter File
Select the Parameter file which contains the loading information.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar Elements
The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

Note

If boundary conditions are defined for certain entities within the model, the Defined elements should be selected for the appropriate element type. For example, if pressures are defined on surface elements, and **Shell Elements** is set to **None**, then the pressures applied to the surface elements will be ignored.

Thickness Distribution
Select from **BC Defined**, **Distributed**, or **Averaged**.

The **Advanced** Edit option has the additional options.

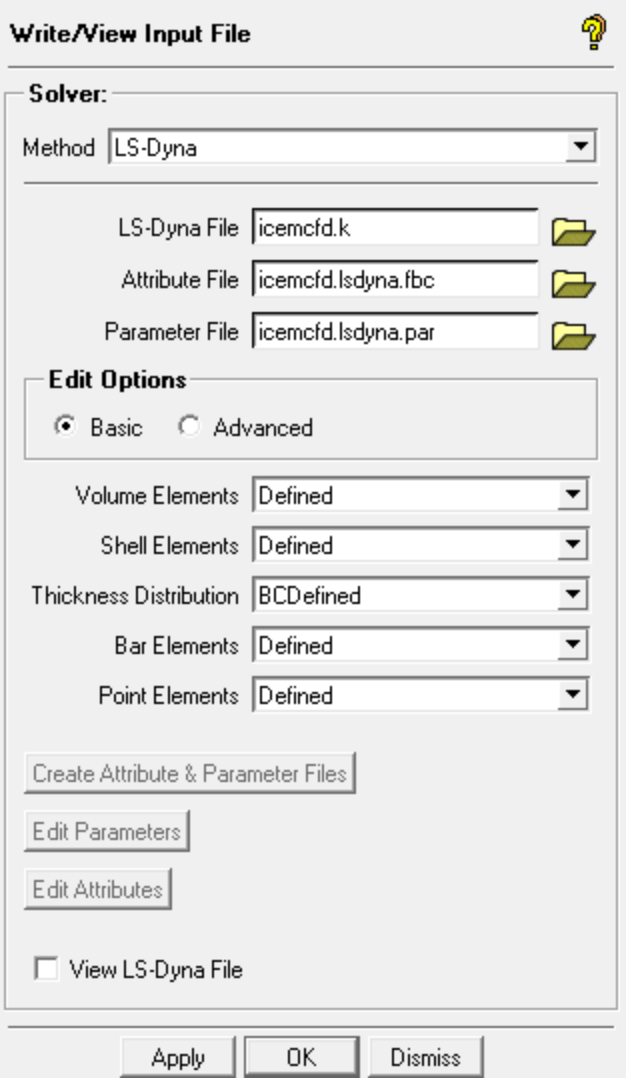
Create Attribute & Parameters Files
This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided. The information about these files can be obtained on the Parameters file or Attributes file pages.

With the help of these files, the translator will write an ANSYS input file whenever **Apply** is pressed.

View ANSYS File
If this option is enabled, then after the ANSYS input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LS-DYNA Write/View Input File



LS-DYNA File
Select the LS-DYNA input file. It is similar to the Nastran *.dat file.

Attribute File
Select the Attribute file which contains information about all the parts and material properties.

Parameter File
Select the Parameter file which contains the loading information.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar, Point Elements
The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include the elements.

Thickness Distribution
Select from **BC Defined**, **Distributed**, or **Averaged**.

The **Advanced** Edit option has the additional options.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Attribute & Parameters Files

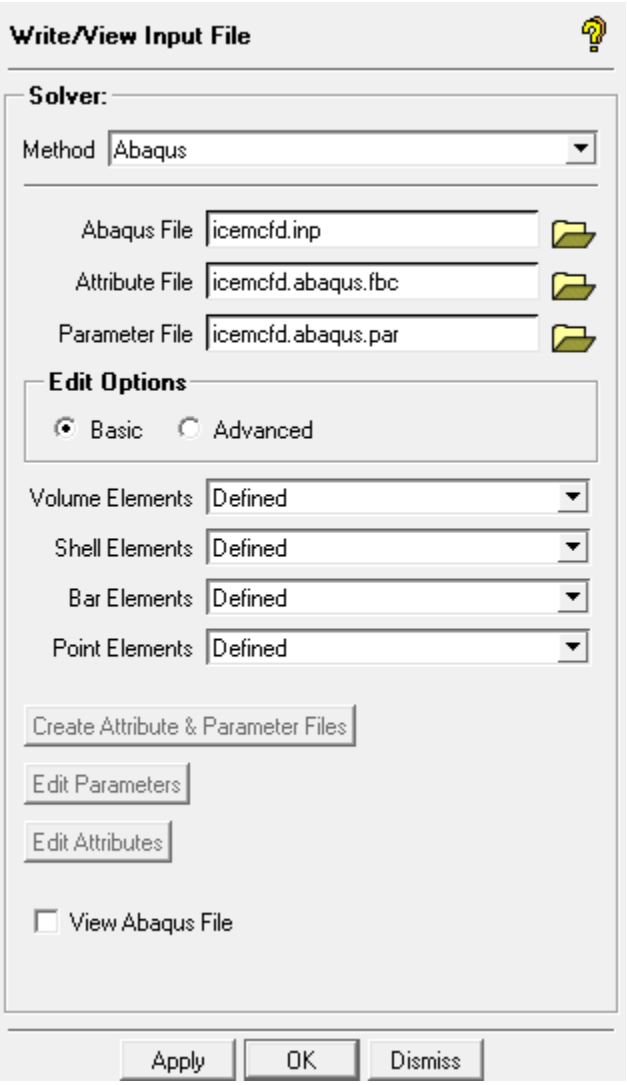
This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided.

The **Edit Attributes** button opens the **Family Boundary Condition** window, where loads and other boundary conditions can be defined.

View LS-DYNA File

If this option is enabled, then after the LS-DYNA input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

Abaqus Write/View Input File



Abaqus File

Select the Abaqus input file. It is similar to the Nastran *.dat file.

Attribute File

Select the Attribute file which contains information about all the parts and material properties.

Parameter File

Select the Parameter file which contains the loading information.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are two Edit Options: **Basic** and **Advanced**. Both options contain the following options.

Volume, Shell, Bar, Point Elements
The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include any elements.

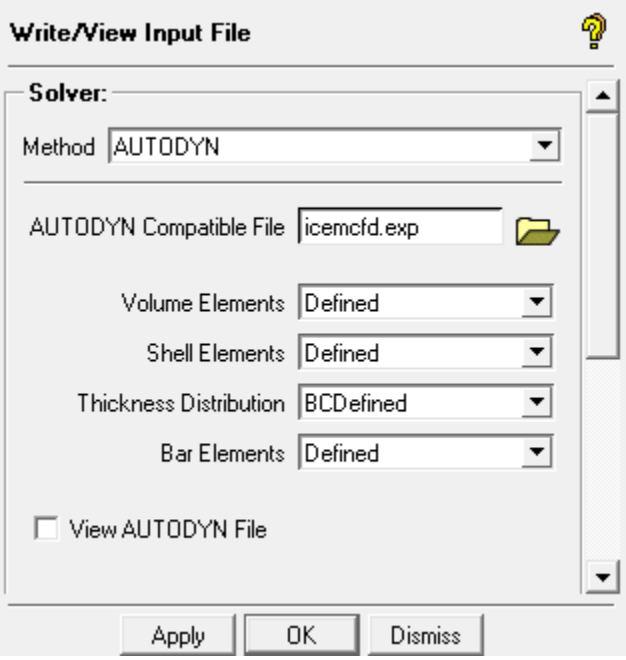
Thickness Distribution
Select from **BC Defined**, **Distributed**, or **Averaged**.

The **Advanced** Edit option has the additional options.

Create Attribute & Parameters Files
This option will generate the Parameter and Attribute files in the project directory. You can edit the data of these files using the separate **Edit Parameters** and **Edit Attributes** buttons provided.

View Abaqus File
If this option is enabled, then after the Abaqus input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

Autodyn Write/View Input File



Autodyn Compatible File
Select the Autodyn Compatible File (*.k).

Volume, Shell, Bar Elements
The elements which are to be included in the input file. Select **Defined** to include only the defined elements, **All** to include all available elements, or **None**, to not include the elements.

Thickness Distribution
Select from **BC Defined**, **Distributed**, or **Averaged**.

View Autodyn File
If this option is enabled, then after the Autodyn input file is written, it will be opened in the default editor. The file can be edited at this time, before it is run.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Submit Solver Run

Use this option to initiate the Solver Run.

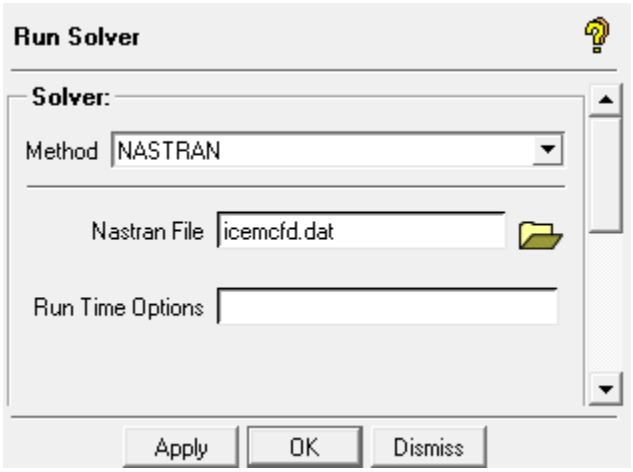
The details and choices for each Solver **Method** are described in the following sections.

NASTRAN Submit Solver Run
ANSYS Submit Solver Run
LS-DYNA Submit Solver Run

Note

- Abaqus solver run has not been implemented.
- The Autodyn solver can be run in the Workbench environment.

NASTRAN Submit Solver Run

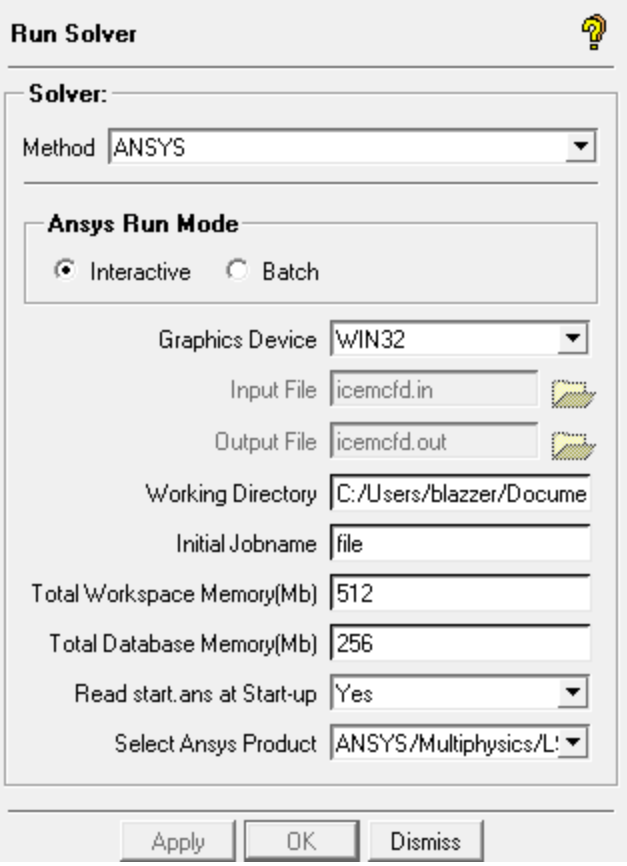


Nastran File
Select the required Nastran input file (*.dat).

Runtime Options
The maximum time allowed for the Nastran run.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS Submit Solver Run



ANSYS Run Mode

There are two choices for the run mode, **Interactive** and **Batch**.

Graphics Device

For Interactive Mode, the GUI requires a terminal that supports graphics.

Input and Output File

If Batch Mode is selected, you need to specify the Input and Output files. The Input file contains the ANSYS commands submitted for batch execution. The Output file contains the text output by the program. If the supplied file name already exists in the working directory, then it will be overwritten when the batch job is started.

Working Directory

The directory in which the ANSYS run will be executed. Files that are written are created in this directory.

Initial Jobname

The base file name used for all files generated by the ANSYS run.

Total Workspace Memory (Mb)

The amount of memory requested for the ANSYS run.

Total Database Memory (Mb)

The portion of total memory that the database will use.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

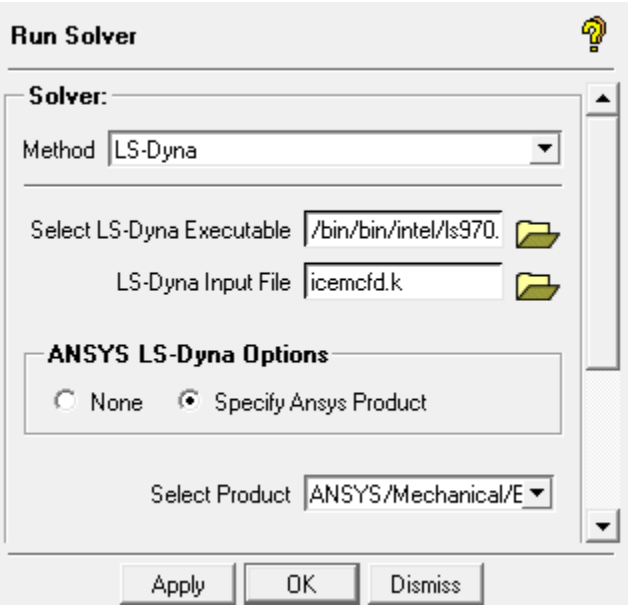
Read start.ans at Start-up

To choose whether to read the start.ans file. ANSYS users can include commands to be executed at start-up in this file.

Select ANSYS Product

Select from the list of ANSYS products that are licensed and available.

LS-DYNA Submit Solver Run



Select LS-DYNA Executable

Select the LS-DYNA executable file.

LS-DYNA Input File

The LS-DYNA k file, written in the previous step as the input for the solver.

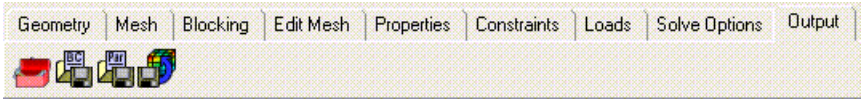
ANSYS LS-DYNA Options

If you want to use an ANSYS license for running the LS-DYNA solver, select the applicable ANSYS license product.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Output

Figure 497: Output Menu



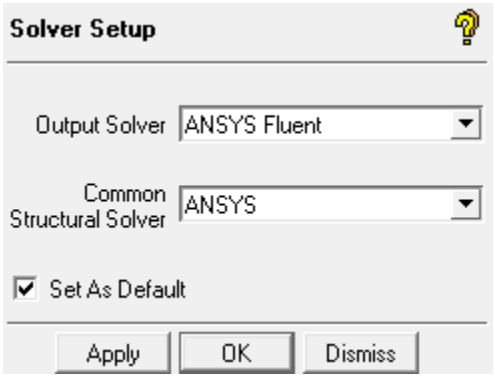
ANSYS ICEM CFD allows you to write input files for many different **Output Solvers**. The Output Menu allows you to choose between different solvers and specify the file related options and settings.

Select Solver
Boundary Conditions
Edit Parameters
Write Input

Select Solver

The **Select Solver** option will open a window to select an **Output Solver** (CFD) or **Common Structural Solver** (FEA).

Figure 498: Select Solver DEZ



Note

Both FEA and CFD properties can be set for the same model.

ANSYS ICEM CFD allows you to write input files for any of the following **Output Solvers**:

Table 9: Supported Output Solvers

ABAQUS	ACRi	ACTRAN	ACUSOLVE	ADH_NS
ADINA	ALPHA-FLOW	ANSYS	ANSYS CFX	ANSYS Fluent
AUTOCFD	AcFlux	Airflo3D	Attila	BAGGER
C-MOLD	CEDRE	CFD++	CFDRC-DTF	CFL3D

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFX-4	CFX-TASCflow	CGNS	CHAD	CMG_STARS
COBALT	COLISEUM	CRSOL	CRUNCH CFD	CSP
Comco	Common Structur-al	Concert3D	Datex	EM
EXODUS-II	FANSC	FASTEST-3D	FENFLOSS	FENSAP
FIELDVIEW	FLEX	FastU	Fidap	Fire_V7
Fire_V8	Flow-Logic	FlowCart	Fluent V4	GSMAC-DF
HDF (GE)	HP	IBM-Bem	ICAT	ICU
IDEAS	IMPNS	iPLES	KIVA-3(V)	KIVA-4
LAURA	LL-DYNA3D	LS-DYNA3D	MACS	MAGREC
MAZe	MOUSE	Multiblock-info	N3S-Natur	NCC
NOPO	NPARC	NSU3D	Nastran	Ns3D
P&W Common	PAB3D	PARC	PHOENICS	PMARC
POPINDA	PRECISE (Imperial)	PRECISE (SI)	Patran	Plot3D
Poly3D	Polyflow	RADIOSS	RAVEN	RTT
RadTherm (.tdf)	SAUNA	SC/Tetra	SCRYU	SPECTRUM-CENT-RIC
SPLITFLOW	STAR-4	STAR-CCM+	STAR-CD 3.1.0/3.0.5	STAR-CD 3.2.0
STARS	STL	SpecElem	TGrid	TLNS3D-mb
TNO	TSAR	Team	Tranair	Trio-U
UGRID	UH3D	USA	USM3D	USMKV3V
VRML	VSAERO/USAERO	VULCAN	Vectis	WIND
WIND-MASTER	ZEN			

Note

- For Fluent V6, use ANSYS Fluent.
- CFD++ may not be able to import part names longer than 12 characters. This is a CFD++ limitation.

Once a solver is selected, a window will open with options relating to that solver. You can then modify the solver parameters, and write the input files. If configured properly, you can also run the solver directly from ANSYS ICEM CFD.

Note

The output interfaces are separate executables that operate on project files that have been saved to disk. This means you will need to save the project, particularly the mesh, family boundary conditions, and/or properties files before sending output to the solver. If your mesh is large and you are concerned about memory limitations and do not want the mesh loaded twice into memory (the editor and the output interface), you can then unload the mesh and proceed with the output based on the saved mesh file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

More information about the ANSYS ICEM CFD Output Interfaces is available from the **Help** menu. The **Output Interfaces** option opens the ANSYS ICEM CFD Output Interfaces information in a browser. For information about a specific interface, refer to the Table of Supported Solvers and click the name of the interface.

For more information on the **Common Structural Solvers**, see the Solve Options chapter.

Some of the commonly used output interfaces are:
ANSYS
ANSYS CFX
ANSYS Fluent
CGNS
Polyflow

ANSYS

Solver	ANSYS
Type of mesh	unstructured mesh
Dimension	2D and 3D

This interface writes the command deck or batch file according to the ANSYS 5.7 specifications. The ANSYS batch file created with this translator contains the following sections:

- Element type (ET)
- Real constants (R/RMORE)
- Material properties (MP, MPDATA/MPTEMP)
- Local coordinate systems (CSKP)
- Unblocked option - Node coordinates (N), Element connectivity table (TYPE, REAL, MAT, E/EMORE), Components (CM)
- Blocked option - Node coordinates (NBLOCK), Element connectivity table (EBLOCK), Components (CMBLOCK)
- Nodal DoF constraints (D)
- Coupled degrees of freedom (CP)
- Constraint equations (CE)
- Nodal surface loads (SF)
- Uniformly distributed forces over surfaces (SURF154/SFE)
- Nodal body loads (BF)
- Nodal forces (F)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

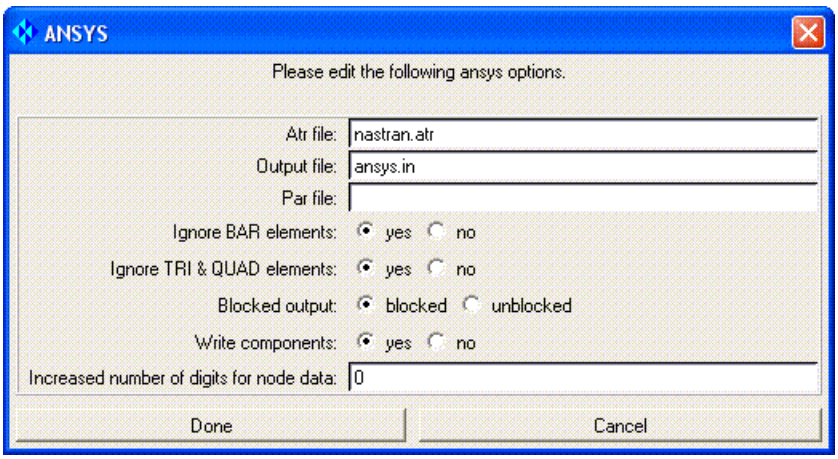
Creating the ANSYS Input File

The translator writes the ANSYS batch file using the following files:

- unstructured domain file (supports both linear & quadratic meshes)
- boundary condition file (.fbc)
- solver parameter file (.par)

The ANSYS output interface can be invoked from the ANSYS ICEM CFD main window. Select **ANSYS** in the **Output Solver** drop-down list and click **Apply**. Then select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the ANSYS format. An existing unstructured mesh file (*.uns) must first be selected.

Figure 499: ANSYS Options



Specify the following:

- the boundary condition file (*.atr)
- the solver parameter file (*.par)
- the ANSYS batch input file (*.in)
- whether to ignore BAR (1-D) elements (default: yes)
- whether to ignore TRI & QUAD (2-D) elements (default: yes)
- specify whether to output components (default: yes)
- specify output format for nodes, elements & components data (default: blocked)

Defining Solver Parameters

Select Mesh order:

- Low order / Linear (default)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- High order / Quadratic

Note

For meshes to have quadratic elements such as ANSYS element 95 (for Structural analysis) or element 119 (for HF Electromagnetic analysis), first perform the Linear-> Quadratic operation on the mesh in order to create the higher order elements.

Set ANSYS Analysis Options:

1. Select ANSYS Product

   The following are currently available:

   - Structural Analysis (ST - default)
   - High Frequency Electromagnetic Analysis (EH)
   - Flotran CFD Analysis (FL)
   - Thermal Analysis (TH)

2. Select Analysis type (ANTYPE)

   The default analysis type is None, for which the solution preprocessor (/SOLU) and subsequent commands are not output. The options for various analyses as follows:

   Structural analysis:

   STATIC

   - Equation solvers (EQSLV): Direct (SPARSE), Pre-conditioned Conjugate Gradient iterative (PCG)
   - Include or exclude large deformation effects (NLGEOM)
   - Specify the number of substeps (NSUBST)

   MODAL

   - Mode extraction methods (MODOPT): Block Lanczos (LANB)
   - Number of modes to expand and write (MXPAND)

   Thermal analysis:

   STATIC

   - Equation solvers (EQSLV): Incomplete Cholesky Conjugate Gradient iterative (ICCG)
   - Include or exclude large deformation effects (NLGEOM)
   - Specify the number of substeps (NSUBST)

3. Set number of processors (/CONFIG,NPROC)
4. Select Exit option for ANSYS batch job (/EXIT)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If set to **Yes**, the /EXIT command with the selected sub-option is written at the end of the ANSYS batch input file. If **No** (default), the exit command is not written.

Set Flotran Options:

1. Specify fluid families:

   Select from **All**, **None** or specify with **Create new** option. The elements of all the specified fluid families will be automatically assigned material id 1.

2. Set solution algorithms (FLDATA1):

   Set On(TRUE) or Off(FALSE) flags for the Flotran solution algorithms. If the default setting is changed, the FLDATA1 command appears in the ANSYS batch file.

3. Set convergence monitors for the DoF's (FLDATA3)

4. Set output & Storage controls (FLDATA5)

   The FLDATA5 commands appear as per the logic for (2).

5. Set convergence monitor output controls (FLDATA6)

6. Define fluid properties (FLDATA7-FLDATA11)

   Any number of fluid properties can be created as required. The values required for FLDATA8-FLDATA11 can be entered for all properties other than Air properties.

7. Set property variation flags (FLDATA13)

   If a property has been defined in (6) and is set as varying, the FLDATA13 command appears in the ANSYS batch file.

8. Specify reference properties (FLDATA14-FLDATA17)

Define Global Parameters: The global loads supported are: acceleration (ACEL), rotational velocity (OMEGA) and rotational acceleration (DOMEGA).

Define Real Constant sets (R/RMORE): Any number of named Real Constant sets can be created as required. The names can be associated either with the default elements or later, with a family in the b.c. menu. ID values are generated internally for the named sets, which appear in the batch file. Once a set is created (R), six values can be input for each subset. There could be more than one subset for a set. The additional subsets are written out with the RMORE command.

Set Element defaults: Specify defaults for 2-D & 3-D elements for the ANSYS product & Element order, selected above. The following ANSYS elements are currently supported:

ANSYS product	Element Type(s)	Supported Elements			
		Low order (Linear)		High order (Quadratic)	
		2-D	3-D	2-D	3-D
Structural	SHELL/SOLID	41, 43, 63, 143, 181*	45, 64, 65, 185 (default)	93*	92, 95, 1110 (default), 187
HF Emag	HF	--	--	--	119, 120, Auto** (default)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flotran CFD	FLUID	141 (default)	142(default)	--	--
Thermal	SOLID	--	--	--	87, 90, Auto**(default)

* The default ET for these element types is "None", in which case the elements for the specified type of ICEM CFD families are not output. However, nodal components are still output if user selects not to ignore 2-D (TRI & QUAD) elements.

** These get automatically mapped to the suitable ANSYS element depending on elements of the underlying ICEM CFD family.

The following default values are provided for each of the tabulated element:

1. Real constant set name: Select from the list of Real Constant sets created earlier. (default - None)
2. Key options (KOP1-6)
3. Solution printout flag (INOPR)

Notes

1. The element defaults specified in this section can be overridden by specifying different Element type through the boundary conditions menu.
2. The 1-D elements for Beams, Spars and Springs & Dampers can be specified through the b.c. menu. The 1-D elements currently supported as follows:

ANSYS 1-D Element Type	Supported Elements
Spars (LINK)	8, 10, 11
Beams (BEAM)	4, 24, 44
Springs & Dampers (COMBIN)	7, 14, 37, 39, 40

Define temperature dependant property data (MPDATA/MPTEMP): Any number of Property-Temperature named data sets can be created as required. The names can be later associated with a family in the b.c. menu. Under a given named data set, one can create as many Temperature-Property value pairs as required.

Note

The temperature values keyed in the data points should be in non-descending order.

Define coupled DOF's (CP): The coupled DOF's can be applied to the nodes of one or more families. The CP command is issued as follows:

```
CP, NEXT, LAB, fam1.nid1
CP, HIGH, LAB, fam1.nid2
...
CP, HIGH, LAB, fam1.nidlast
CP, HIGH, LAB, fam2.nid1
```

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
...
CP, HIGH, LAB, fam2.nidlast
...
CP, HIGH, LAB, famn.nid1
...
CP, HIGH, LAB, famn.nidlast
```

pre-set variables:

- fam1.nid1, fam1.nid2, ... , fam1.nidlast, fam2.nid1, ... , fam2.nidlast, famn.nid1, ... , famn.nidlast = Node ID's of 'n' families specified by user

user defined variables:

Select the DOF label for CP, LAB, see list below.
Select families, fam1 through famn.

If the ANSYS interface option "output components" is "yes", the CP command is issued in the following steps:

```
CMSEL, S, fam1
CMSEL, A, fam2
...
CMSEL, A, famn
CP,NEXT,LAB,ALL
```

The CP command can be repeated for the same or different labels as many times as required, within the ANSYS limitation that the selection of families across all sets is kept mutually exclusive. If not, a warning is issued. The DOF labels for CP available within the interface are:

```
Analysis type    Label
-------------    -----

Structural       ALL, UX, UY, UZ, ROTX, ROTY, ROTZ
Thermal          ALL, TEMP
Emag-HF          ALL, AX
Flotran          Not supported
```

Note

The choice lists for the selection of families appear in the solver parameters menu ONLY IF the boundary condition file is loaded earlier.

Define constraint equations (CE): The constraint equations can be applied either to the nodes of a single family or to a pair of families that contain periodic nodes. The CE command when applied to a single family is issued as follows:

```
CE, NEXT, CONST, fam1.nid1, Lab1, C1, fam1.nid1, Lab2, C2, fam1.nid1, Lab3, C3
CE, HIGH, CONST, fam1.nid1, Lab4, C4, fam1.nid1, Lab5, C5, fam1.nid1, Lab6, C6
CE, NEXT, CONST, fam1.nid2, Lab1, C1, fam1.nid2, Lab2, C2, fam1.nid2, Lab3, C3
CE, HIGH, CONST, fam1.nid2, Lab4, C4, fam1.nid2, Lab5 C5, fam1.nid2, Lab6, C6
...
CE, NEXT, CONST, fam1.nidlast, Lab1, C1, fam1.nidlast, Lab2, C2, fam1.nidlast, Lab3, C3
CE, HIGH, CONST, fam1.nidlast, Lab4, C4, fam1.nidlast , Lab5, C5, fam1.nidlast, Lab6, C6

If the periodic family is specified, the CE command output appears as follows:
CE, NEXT, CONST, fam1.nid1, Lab1, C1, fam1.nid1, Lab2, C2, fam1.nid1, Lab3, C3
CE, HIGH, CONST, fam1.nid1, Lab4, C4, fam1.nid1, Lab5, C5, fam1.nid1, Lab6, C6
CE, HIGH, CONST, fam2.nid1, Lab1, K1, fam2.nid1, Lab2, K2, fam2.nid1, Lab3, K3
CE, HIGH, CONST, fam2.nid1, Lab4, K4, fam2.nid1, Lab5, K5, fam2.nid1 Lab6, K6
...
```

pre-set variables:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- fam1.nid1, fam1.nid2, ... , fam1.nidlast = Node ID's of the first family specified by user
- fam2.nid1, fam2.nid2, ... , fam2.nidlast = Node ID's of the periodic family specified by user

user defined variables:

Selected first family, fam1
Second periodic family, if specified, fam2
Enter the value of equation constant, CONST (real)
Selected DOF labels for CE, Lab1 through Lab6, see list below
Enter the values of DOF multipliers for Lab1 through Lab6, for fam1, C1 through C6 (real)
Enter the values of DOF multipliers for Lab1 through Lab6, for fam2, K1 through K6 (real)

The CE command can be repeated as many times as required. Currently no checks are carried out to check if the DOF's appearing in the CE commands are duplicated in any D or CP commands issued. The DOF labels for CE available within the interface are:

```
Analysis type    Label
-------------    -----

Structural       Label1=UX, Label2=UY, Label3=UZ, Label4=ROTX, Label5=ROTY, Label6=ROTZ
Thermal          Label1=TEMP
Emag-HF          Label1=AX
Flotran          Not supported
```

Notes

1. The choice lists for the selection of families appear in the solver parameters menu ONLY IF the boundary condition file is loaded earlier.
2. When the constraint equations between nodes of a pair of families are to be defined, the ICEM CFD menu, Geometry>Mesh Params>Define periodicity should be used. The interface then automatically finds the corresponding periodic nodes in these two families. In this case the constraint equations are defined only for such nodes. If no periodic nodes are found, a warning is issued.

Defining Boundary Conditions for ANSYS

This section describes how to define ANSYS commands using the **Output > Boundary Conditions** menu. Refer to the General Remarks section for more details

Note

The left pane of the menu displays families grouped according to the element dimensions. If any families appear under the type **Mixed/unknown**, these families should be split into more families so that each new family contains elements having same dimension. Alternatively, the choices of writing 1-D & 2-D elements can be suitably made in the write input menu.

Element Type (ET): The ET command contains the following:

ET, ITYPE, ENAME, KOP1, KOP2, KOP3, KOP4, KOP5, KOP6, INOPR
user defined variables:

Select the value of ELEMENT TYPE (integer)
Select the value of KOP1 (integer)

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the value of KOP2 (integer)
Select the value of KOP3 (integer)
Select the value of KOP4 (integer)
Select the value of KOP5 (integer)
Select the value of KOP6 (integer)
Select the value of INOPR (integer)
Select the name of Real Constant Set* (string)

The ET command needs to be specified for each family defined in the boundary condition menu. It will automatically apply to all elements of these families. The element type number correspond to the number following the element name as given in the ANSYS element library (e.g., 45 would refer to SOLID45 element type). ITYPE values are internally generated by the interface.

The element types are linked to the elements with the ANSYS command TYPE when written in unblocked format. When written in blocked format, element type is specified as part of the EBLOCK command.
The real constant sets are linked to the elements with the ANSYS command REAL when written in unblocked format. When written in blocked format, real set id is specified as part of the EBLOCK command.

Material Property (MP, MPDATA/MPTEMP): The MP command contains the following:

MP, LAB, MAT, C0, C1, C2, C3, C4

user defined variables:

Select the label (LAB), see list below.
Property-Temperature data option "None"
Enter the value of C0 (real)
Enter the value of C1 (real)
Enter the value of C2 (real)
Enter the value of C3 (real)
Enter the value of C4 (real)

The temperature dependant material properties can be specified with the same menu, by using the "Specify" option.

The MPDATA/MPTEMP commands contain the following:

MPDATA, LAB, MAT, STLOC, C1, C2, C3, C4, C5, C6
MPTEMP, STLOC, T1, T2, T3, T4, T5, T6

pre-set variables:

- ***Material id(MAT) = ICEM CFD family ID.

  *** **For Flotran analyses**:

  Material id(MAT) = ICEM CFD family id + 1, for non-fluid families, and = 1, for fluid families.

- STLOC

user defined variables:

Select the label (LAB), see list below.
Property-Temperature data option "Specify"
Select the name of Property-Temperature Data Set* (string)

The valid material property labels are:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Label
Elastic moduli	EX, EY, EZ
Coefficients of thermal expansion	ALPX, ALPY, ALPZ
Reference temperature	REFT
Major Poisson's ratios	NUXY, NUYZ, NUXZ
Minor Poisson's ratios	PRXY, PRYZ, PRXZ
Shear moduli	GXY, GYZ, GXZ
K matrix multiplier for damping	DAMP
Coefficient of friction	MU
Mass density	DENS
Specific heat	C
Enthalpy	ENTH
Thermal conductivities	KXX, KYY, KZZ
Convection or film coefficient	HF
Emissivity	EMIS
Heat generation rate	QRATE
Viscosity	VISC
Sonic velocity	SONC
Electrical resistivities	RSVX, RSVY, RSVZ
Electrical conductivities**	CNDX, CNDY, CNDZ
Electrical permitivities	PERX, PERY, PERZ
Magnetic relative permeabilities	MURX, MURY, MURZ
Magnetic coercive forces	MGXX, MGYY, MGZZ
Loss tangent	LSST

The material properties are linked to the elements with the ANSYS command MAT when written in unblocked format. When written in blocked format, material id is specified as part of the EBLOCK command.

*Ref: "Define temperature dependant property data" section for defining solver parameters.
** CNDn = 1/RSVn, RSVn values appear in the ANSYS batch file.

Local coordinate system (CSKP): The CSKP command is written along with a fixed set of other commands as follows:

CSYS, 0
K,1,x1,y1,z1
K,2,x2,y2,z2
K,3,x3,y3,z3
CSKP,KCN,KCS,1,2,3
KDELE,1,3,1

pre-set variables:

Local Coordinate Systems are created using View/Coord systems/Define menu of MED.

The pre-set variable values are set as follows:

- KCN = Arbitrary reference number (>10) assigned to the coordinate system.
- KCS = Coordinate system type, as specified in MED.
- x_n through z_n : Internally calculated from coordinates of the origin and the two axes, as specified in MED.

user defined variables:

Select the name of Local Coordinate System* (string)
*As defined in MED.

When a family is associated with a Local Coordinate system, all the nodes of elements in the family are aligned with this system with CSYS, KCN and NROTAT commands before applying nodal DoF constraints

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(D), nodal body loads (BF) and nodal forces (F). If any two families which have common nodes are associated with different local coordinate systems, a warning is issued.

Nodal DoF Constraints (D): Before a D command is issued, the DOF accumulation flag is set for addition with

DCUM, ADD, 1.0, 1.0, 0.0

The D command contains the following:

NSEL, ALL
D, NODE, LAB, VALUE, VALUE2, NEND, NINC, Lab2, Lab3, Lab4, Lab5, Lab6

pre-set variables:

- NODE = ...each node of the family.
- NEND, NINC, Lab2, Lab3, Lab4, Lab5, Lab6: not used

user defined variables:

Select the degree of freedom label (LAB), see list below
Enter the degree of freedom value (real)
Enter the imaginary part of the DoF value2, if applicable (real)

If the ANSYS interface option "output components" is "yes", the D command is issued in the following steps:

CMSEL, S, family_name
D, ALL, LAB, VALUE, VALUE2, NEND, NINC, Lab2, Lab3, Lab4, Lab5, Lab6

The valid degree of freedom labels are:

Description	Label
Use all appropriate labels	ALL
Structural labels:	
displacements	UX, UY, UZ
rotations	ROTX, ROTY, ROTZ
Thermal labels:	
temperature	TEMP
Fluid labels:	
pressure	PRES
velocities	VX, VY, VZ
turbulent kinetic energy	ENKE
turb.kin. E dissipation rate	ENDS
Electric labels:	
voltage	VOLT
Magnetic labels:	
scalar magnetic potential	MAG
vector magnetic potentials	AX, AY, AZ

Nodal Surface Loads (SF): The SF command contains the following:

```
ESEL, ALL
NSEL, S, NODE, ,nid1,nid1
NSEL, A, NODE, nid2, nid2
...
SF, ALL, LAB, VALUE, VALUE2
```

pre-set variables:

- nid1, nid2 ... = Node ID's of the family

user defined variables:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the surface load label (LAB), see list below
Enter the surface load value (real)
Enter the surface load value2, if applicable (real)

If the ANSYS interface option "output components" is "yes", the SF command is issued in the following steps:

ESEL, ALL
CMSEL, S, *family_name*
SF, ALL, LAB, VALUE, VALUE2

The valid surface load labels are:

```
Description                                 Label
-----------                                 -----
Structural labels:
    pressure                                PRES
Thermal labels:
    convection                              CONV
    heat flux                               HFLUX
    radiation                               RAD
Fluid labels:
    constant total pressure                 PTOT
Electric labels:
    surface charge density                  CHRGS
Magnetic labels:
    magnetic circuit interface              MCI
HF Emag labels:
    exterior surface flag                   INF
    exterior waveguide port                 PORT
    surface shielding properties            SHLD
    impedance                               IMPD
```

The SF command applies to 2-D or 3-D families only. The "ESEL, ALL" command is issued first, since SF command requires the elements to be exposed before the application.

The SFE command contains the following:

```
ESEL, ALL
NSEL, S, NODE, ,nid1,nid1
NSEL, A, NODE, nid2, nid2
...
MAT, mid
ET, eid, 154
! Keyopt for normal pressure
KEYOPT, eid, 6, kop6
! Keyopts for directional pressure
KEYOPT, eid, 11, kop11
KEYOPT, eid, 12, kop12
TYPE, eid
ESURF
ESEL,S,MAT,,mid,,,
! For normal pressure
SFE, ALL, 1, PRES, , (Total Force)/family_area
! For directional pressure
SFE, ALL, 5, PRES, , (Total Force)/family_area, Px, Py, Pz
```

Uniformly distributed forces over surfaces (SURF154/SFE): The interface uses overlaid SURF154 elements in combination with SFE command to apply uniformly distributed forces over surfaces. This command is only valid in the structural analyses. The real constant set number and element coordinate system are set to zero before the subsequent ESURF command that generates the overlaid elements.

The SFE command contains the following:

```
ESEL, ALL
NSEL, S, NODE, ,nid1,nid1
NSEL, A, NODE, nid2, nid2
...
```

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
MAT, mid
ET, eid, 154
! Keyopt for normal pressure
KEYOPT, eid, 6, kop6
! Keyopts for directional pressure
KEYOPT, eid, 11, kop11
KEYOPT, eid, 12, kop12
TYPE, eid
ESURF
ESEL,S,MAT,,mid,,,
! For normal pressure
SFE, ALL, 1, PRES, , (Total Force)/family_area
! For directional pressure
SFE, ALL, 5, PRES, , (Total Force)/family_area, Px, Py, Pz
```

pre-set variables:

- nid1, nid2 ... = Node ID's of the family
- mid = Unique material ID for the SURF154 elements generated with the ESURF command
- eid = Unique element type ID for the SURF154 elements
- family_area = Total surface area of the elements in the family

user defined variables:

Enter the surface Total Force value (real)
Select key option for SURF154 elements for normal pressure, kop6 (int)
Select key options for SURF154 elements for directional pressure, kop11/12 (int)
Enter the vector for directional pressure, "Px Py Pz" (real real real)

If the ANSYS interface option "output components" is "yes", the set of NSEL commands are replaced by:

CMSEL, S, *family_name*

The valid surface load labels are:

```
Description                                Label
-----------                                -----
Structural labels:
    pressure                               PRES
```

Nodal Body Loads (BF): The BF command contains the following:

NSEL, ALL
BF, NODE, LAB, VAL1, VAL2, VAL3, PHASE

pre-set variables:

- NODE = ...each node of the family.

user defined variables:

Select the body load label (LAB), see list below
Enter the body load value1 (real)
Enter the body load value2, if applicable (real)
Enter the body load value3, if applicable (real)
Enter value for phase, if applicable (real)

If the ANSYS interface option "output components" is "yes", the BF command is issued in the following two steps:

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMSEL, S, *family_name*
BF, ALL, LAB, VAL1, VAL2, VAL3, PHASE
The valid body load labels are:

```
Description                                 Label
-----------                                 -----
Structural labels:
    temperature                             TEMP
    fluence                                 FLUE
Thermal labels:
    heat generation rate                    HGEN
Electric labels:
    charge density                          CHRGD
    temperature                             TEMP
Magnetic labels:
    magnetic virtual displacement flag      MVDI
    temperature                             TEMP
HF Emag labels:
    current density                         JS
    interior port                           PORT
    magnetic field                          H
    electric field                          EF
Flotran labels:
    force density in momentum equation      FORC
    heat generation rate                    HGEN
```

Nodal Forces (F): The F command contains the following:

NSEL, ALL
F, NODE, LAB, VALUE, VALUE2, NEND, NINC
pre-set variables:

- NODE = ...each node of the family.
- NEND, NINC: not used

user defined variables:

Select the force label (LAB), see list below
Enter the force value (real)
Enter the imaginary part of the force value2, if applicable (real)
If the ANSYS interface option "output components" is "yes", the F command is issued in the following steps:

CMSEL, S, *family_name*
F, ALL, LAB, VALUE, VALUE2
The valid force labels are:

```
Description                                 Label
-----------                                 -----
Structural labels:
    forces                                  FX, FY, FZ
    moments                                 MX, MY, MZ
Thermal labels:
    heat flow                               HEAT
Fluid labels:
    fluid flow                              FLOW
Electric labels:
    electric charge                         CHRG
    current flow                            AMPS
Magnetic labels:
    magnetic current segments               CSGX, CSGY, CSGZ
    magnetic flux                           FLUX
Flotran labels:
    forces                                  FX, FY, FZ
```

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS CFX

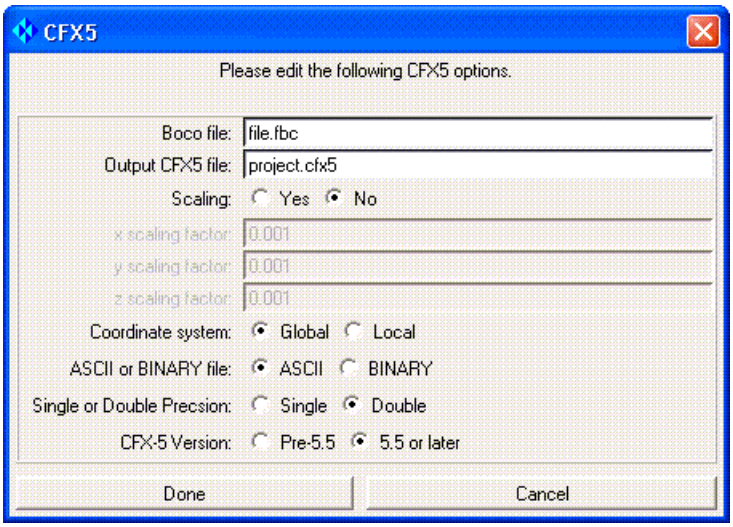
Solver	ANSYS CFX (CFX–5)
Type of mesh	unstructured mesh
Dimension	3D

This translator writes coordinates, connectivity and family information of an unstructured mesh file in ANSYS CFX (CFX-5.2) format.

Creating the ANSYS CFX (CFX-5.2) Input File

The translator writes the ANSYS CFX (CFX-5.2) input file using an unstructured mesh file and its boundary condition file. To create the ANSYS CFX (CFX-5.2) file, select the translator **ANSYS CFX** in the **Output Solver** list and click **Apply**. Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the ANSYS CFX format.

Figure 500: ANSYS CFX Options



Specify the following:

- the name of the boundary condition file
- the name of the ANSYS CFX (CFX-5) file
- the scaling factor for the coordinates in the geometry file (optional)
- the coordinate system, global or local
- ASCII or binary output
- single or double precision
- the ANSYS CFX (CFX-5) version: **Pre-5.5** or **5.5 or later**

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The interface generates the ANSYS CFX (CFX-5) file in the project directory.

Defining Boundary Conditions

There are no boundary conditions settings required for this translator. Note however that the family information is exported to the ANSYS CFX (CFX-5) file.

ANSYS Fluent

Solver	ANSYS Fluent
Type of mesh	unstructured mesh
Dimension	2D and 3D

The ANSYS Fluent interface creates a Fluent or TGrid mesh file from an unstructured mesh, with the following grid sections:

Section	Index
Comment	0
Dimensions	2
Nodes	10
Cells	12
Faces	13
Periodic Shadow Faces	18
Zone Section	39

The mesh file can be written in ASCII or binary format.

Note

The Fluent output interfaces are separate executables that operate on project files that have been saved to disk. This means you will need to save the project, particularly the mesh, family boundary conditions, and/or properties files before sending output to the solver. If your mesh is large and you are concerned about memory limitations and do not want the mesh loaded twice into memory (the editor and the output interface), you can then unload the mesh and proceed with the output based on the saved mesh file.

Creating the ANSYS Fluent Input File

The translator writes the ANSYS Fluent input file using an unstructured mesh file and its boundary condition file. To create the ANSYS Fluent file, select **ANSYS Fluent** in the **Output Solver** list and click **Apply**. After generating the mesh, boundary conditions can be defined. Supported boundary conditions are:

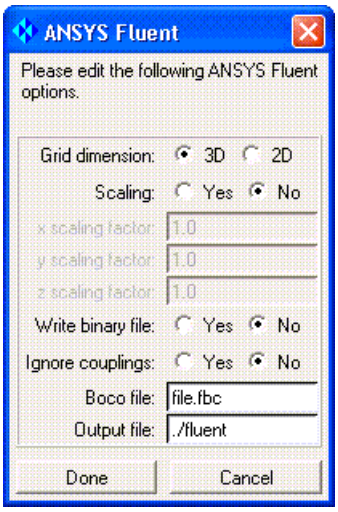
Volume	fluid, porous, solid
Surface	axis, fan, interior, mass-flow-inlet, outflow, periodic, porous-jump, pressure-inlet, pressure-far-field,

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	pressure-outlet, radiator, symmetry, velocity-far-field, velocity-inlet, wall

Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the ANSYS Fluent format.

Figure 501: ANSYS Fluent Options



Specify the following:

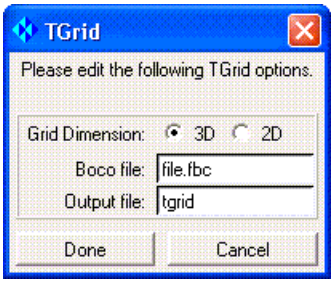
- the grid dimension
- the scaling factor for the coordinates in the geometry file (optional)
- ASCII or binary output
- ignore couplings (yes or no)
- the name of the boundary condition file
- the name of the ANSYS Fluent file

Creating the TGrid Input File

The translator writes the TGrid input file using an unstructured mesh file and its boundary condition file. To create the TGrid file, select **TGrid** in the **Output Solver** list and click **Apply**. Select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the TGrid file.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 502: TGrid Options



Specify the following:

- the grid dimension
- the name of the boundary condition file
- the name of the TGrid file

CGNS

Data Format	CGNS Version 3.1 (ADF / HDF)
Type of mesh	unstructured or structured mesh
Dimension	2D and 3D

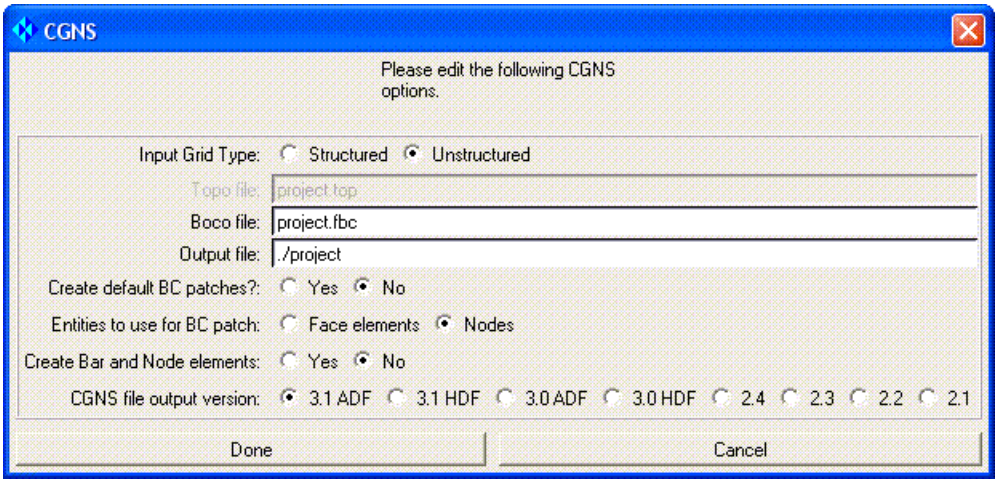
The ANSYS ICEM CFD - CGNS output interface creates a CGNS file in accordance with the SIDS. It uses the CGNS mid-level-library endorsed by the CGNS Steering Committee to create the file. This interface supports both structured and unstructured mesh translation.

Creating the CGNS File

The CGNS output interface can be invoked from the ANSYS ICEM CFD main window. Select **CGNS** in the **Output Solver** drop-down list and click **Apply**. Then select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the CGNS format. An existing unstructured mesh file or a set of structured mesh files must first be selected.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 503: CGNS Options


CGNS
Please edit the following CGNS options.
Input Grid Type: Structured Unstructured
Topo file: project.top
Boco file: project.fbc
Output file: ./project
Create default BC patches?: Yes No
Entities to use for BC patch: Face elements Nodes
Create Bar and Node elements: Yes No
CGNS file output version: 3.1 ADF 3.1 HDF 3.0 ADF 3.0 HDF 2.4 2.3 2.2 2.1
Done
Cancel

In addition, the output interface requires the boundary conditions file, and for structured mesh input, the topology file. It offers options for naming the CGNS file and to select the automatic creation of boundary condition patches. For unstructured mesh input, you have the additional option to select whether the boundary condition patches are defined using nodes or face elements. Default names for the ANSYS ICEM CFD input files and the CGNS file are given. These can be modified as necessary. The interface generates the CGNS file in the current project directory.

Note

When writing the CGNS file, the maximum number of elements in a part multiplied by the number of nodes per element cannot exceed 2 billion.

- For linear Hexa meshes, the maximum number of elements in a part should not exceed 250 million.
- For linear Tetra meshes, the maximum number of elements in a part should not exceed 500 million.

To export meshes with parts comprising a large number of elements, ensure that larger parts are split into smaller parts within these limits.

Connectivity

This translator supports 1-1 connectivity and periodic interfaces.

Defining Boundary Conditions (Optional)

After generating the mesh, and prior to running the translator, boundary conditions can be defined.

Select the **Boundary conditions** option in the **Output** tab. The **Family boundary conditions** window appears, allowing you to set boundary conditions on the mesh families. Two boundary conditions types are available: **BCType** and **BCDataSet**. Note however that **BCDataSet** in a subset of **BCType** in CGNS and therefore cannot be defined by itself. A boundary condition patch can either have a **BCType** alone, or a **BCType** with one or more **BCDataSets**.

BCType

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The boundary condition types (**BCType**) are defined in the SIDS. They identify the equations that should be enforced at a given boundary. To define a boundary condition type, select a boundary family (face in 3D, edge in 2D) and one of the **BCType** options in the provided list.

BCDataSet

Boundary condition data sets are used to define the **BCTypeSimple**, and one or more global boundary condition data. When boundary condition data sets are specified, the boundary condition type must be selected. This can be either **Dirichlet** or **Neumann**. Only uniform boundary condition data are supported by the translator; the data given are applied to the entire boundary condition patch. The list of variables corresponds to the standardized **Data-Name Identifiers** found in Annex A of the SIDS. In general, **BCTypeSimple** is the same as **BCType** and no more than one **BCDataSet** is required. In some particular cases however, the boundary condition type can be flow dependent. In such case, **BCTypeSimple** differs from **BCType**. The SIDS defines these special cases as "compound boundary conditions". For example, an inflow boundary where the flow goes from subsonic to supersonic would require a compound boundary condition. In such case, two **BCDataSets** would be added, each with a different **BCTypeSimple**:

1. with **BCTypeSimple** = **BCInflowSubsonic**
2. with **BCTypeSimple** = **BCInflowSupersonic**

For more details on the use of compound boundary conditions, refer to the SIDS.

3D Elements Groups for Unstructured Grids

The CGNS standard does not provide a specific method for grouping elements. The translator offers this feature by creating a **DataArray_t** node under **Zone_t/DiscreteData_t**. This **DataArray_t** node is named **PID** and contains on array of integers for the pid of each element. Each element's PID is set to its family ID number. The order in which the element's PIDs are listed follows the same order as the global element numbering under **Element_t**.

Polyflow

The interface writes coordinate, connectivity and boundary condition information in the Patran-based Polyflow format. The following Patran packets compose the input file created with this interface:

- Packet 25: Title Card
- Packet 26: Summary Data
- Packet 01: Node Data
- Packet 02: Element Data
- Packet 06: Distributed Loads
- Packet 07: Node Forces
- Packet 99: End of Neutral File

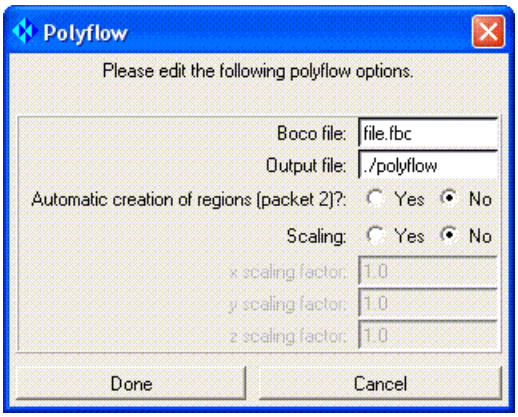
Creating the Polyflow Input File

The translator writes the Polyflow Neutral File Version 2.4 using an unstructured domain file and its boundary condition file. Select **Polyflow** in the **Output Solver** drop-down list and click **Apply**. Then

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

select the **Write input** option in the **Output** tab. A window will open to allow the specification of the files to be translated into the Polyflow format. An existing unstructured mesh file must first be selected.

Figure 504: Polyflow Options



Specify the following:

- the name of the boundary condition file (default is *configuration*/parts_mesh/mesh/boco)
- the name of the Polyflow input file (default is *configuration*/parts_mesh/transfer/*configuration*.poly)
- the scaling factor for the coordinates in the geometry file (optional)

The interface generates the Polyflow file located in the Transfer Shell. You may open a shell window in this directory by selecting the menu items "Utility" and "Transfer Shell".D

Defining Boundary Conditions for Polyflow

In MULCAD the boundary conditions are defined in the "Unspec. Bound. Cond." menu. In the DDN-Mesher Interface, the "general bound. cond." menu is used. Refer to the General Remarks section for more details. For P-CUBE, the NAME field and ID # icon are used to prescribed the Polyflow boundary conditions. Three types of boundary conditions/properties are available for Polyflow: distributed loads, nodes forces and PID numbers.

Distributed Loads:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word DISTR
2.NR            3.integer       ID # icon       Enter the load ID number
```

The element numbers and their load ID are written out in Packet 6. The boundary condition **DISTR** is only recognized on subfaces for a 3D model and on edges for a 2D model.

Node Forces:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word NODE
2.NR            3.integer       ID # icon       Enter the node ID number
```

The node numbers and their load ID are written out in Packet 7. The boundary condition **NODE** is available on subfaces, edges and vertices for a 3D model and on edges and vertices for a 2D model.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element PID:

```
MULCAD          DDN-Mesher      P-CUBE
------          ----------      ------
1.FLAG          1.string1       NAME field      Enter the word PID
2.NR            3.integer       ID # icon       Enter the PID number
```

The element PID are written out in Packet 2. They must be defined on domains for a 3D model and on subfaces for a 2D model. Their default value is 1.

Limitations

It is assumed for the moment that there is no need to define boundary conditions on interfaces since Polyflow expects the load sets to be defined on the boundaries of the domain only.

Polyflow does not allow a mix of 2D and 3D elements in the same model. The type of elements recognized by the Polyflow Translator are:

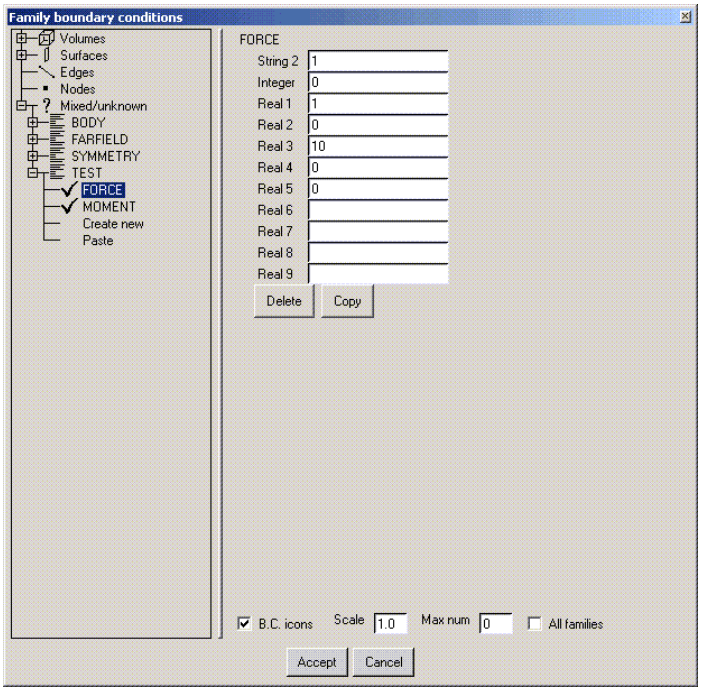
- in 2D:
  - 3 or 6 points triangle
  - 4 or 9 points quadrilateral
- in 3D:
  - 4 or 10 points quadrilateral
  - 8 or 27 points hexahedral
  - 6 or 18 points wedge pentahedral
  - 5 or 14 points pyramidal pentahedral

Boundary Conditions

The **Boundary Conditions** option will open a window to set up and review boundary conditions. Selecting the "+" next to an item will bring down a further list of options and settings. Selecting a part type (Volumes, Surface, Edges, etc.) will display the list of parts of that type in the model. Under each part, the current boundary condition categories for that part are listed, along with the options to create a new boundary condition or paste a copy of one from a different part. The boundary condition categories and types available are specific to the solver. If the Generic solver is selected, no categories will be listed, and instead the boundary condition types will be listed immediately beneath each part. The defined boundary conditions are marked with a check, and selecting it will display the detailed parameters on the right side of the window. At the bottom of the parameters, there are options to **Delete** or **Copy** the parameter values to another boundary condition. A part can contain more than one boundary condition.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 505: Family Boundary Conditions Window



The **Family boundary conditions** window options are as follows:

Volumes
Parts that contain only material points (designated volume region definition) or volumetric elements.

Surfaces
Parts that contain only surfaces or surface elements (tris or quads). The following types of surface parts are defined:

- **One-sided** surfaces or surface elements, such as walls, inlets, and outlets.
- **Two-sided** parts contain zero thickness baffle surfaces or surface elements that have the same volume on either side.
- **Mixed / unknown** parts contain baffles and wall boundaries.

Edges
Parts that contain only curves and/or bar elements.

Nodes
Parts that contain only prescribed points or node entities.

Mixed / unknown
Empty parts or parts that contain a mixture of the geometry or element types listed above.

Create new
This option is available for every boundary condition type, and it opens a window with the list of available boundary condition types specific to the selected solver. Specific parameters for the type can be selected and changed in the parameters list on the right side of the window.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit Parameters

The **Edit Parameters** option opens a window where the solver parameters for the selected solver can be defined. The parameters are organized by type on the left side and the parameter values can be entered in on the right side of the window. Parameter definitions can also be copied and pasted.

Write Input

The **Write Input** option will open a window where you can edit the parameters for the input file. After editing the parameters, you will be prompted to save the project before the input file is written.

ANSYS Input Files

The ANSYS solver supports unstructured mesh.

Note

If boundary conditions are defined for certain entities within the model, the toggle for ignoring the appropriate element type should be disabled. For example, if pressures are defined on surface elements, and **Ignore Shell Elements** is enabled, then the pressures applied to the surface elements will be ignored as well.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSYS ICEM CFD 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.